MACHINE TOOL OPERATION

PART I

Lathe · Bench · Forge

HENRY D. BURGHARDT

MACHINE TOOL OPERATION

Books by

HENRY D. BURGHARDT

MACHINE TOOL OPERATION

PART I

The Lathe, Benchwork and Work at the Forge

420 pages, 5 × 7½, 268 illustrations

PART II

Drilling Machine, Shaper and Planer, Milling and Grinding Machines, Hydraulic Power Transmission, Gears

512 pages, 5 × 7½, 353 illustrations

MACHINE TOOL OPERATION

Part I

THE LATHE

Bench Work and Work at the Forge

BY

HENRY D. BURGHARDT

Formerly, Head of Machine Shops, Wm. L. Dickinson High School, Jersey City, N. J.; Author "Manual for Machinists"; War Department Committee on Education and Special Training

McGRAW-HILL BOOK COMPANY, Inc.

NEW YORK AND LONDON

MACHINE TOOL OPERATION

PRINTED IN THE UNITED STATES OF AMERICA

II

THE MAPLE PRESS COMPANY, YORK, PA.

PREFACE TO THE THIRD EDITION

There are so many occasions when it is necessary to perform simple hand-forging operations in a machine shop that it has seemed worth while to add considerably to the information concerning forge shop tools and processes given in the previous editions.

Beginning on page 347, under the heading *Hand Forging in a Machine Shop*, there will be found many pages of added text and illustrations.

The author appreciates the approval of his work indicated by so many supervisors and teachers, their apprentices and students. It is gratifying to feel that his efforts have been helpful in the training of young men in such important work as machine shop practice.

H. D. BURGHARDT.

PASADENA, CALIF.

PREFACE TO THE FIRST EDITION

Given a class of twenty boys in day school, or twenty men in night school, a subject full of interest, and a teacher full of enthusiasm, can the teacher give information regarding the subject in a "talk" or "lecture," so that the majority of the members of the class will retain it twenty-four hours without some definite reference material?

Certain members of the class may be absent; how are they to get this information except by individual instruction unless a suitable text is provided?

If brief notes concerning the construction of the machine and the various elementary operations are available, will they not serve to shorten the time taken for talks and thus give more time to practical application of principles?

Will not a student have more confidence and hence gain skill more quickly if he has, at hand, the necessary information (and certain information *is* necessary) while performing the operation or studying the mechanism?

The following text is the outgrowth of notes prepared by the author in an effort to answer intelligently the above and many similar questions.

The presentation of this material in book form is the result of requests from many teachers and students that it be given in a more useful and permanent form than that of mimeographed notes. How best to arrange such material is a most difficult problem and any arrangement is open to criticism. This text is planned to permit all possible flexibility in its use. It is not a course of study arranged in any particular pedagogical sequence; the advisability of such a course is doubtful except for the one shop and then only for a given time. Certain chapters comprise a study of operations; other chapters may be used for reference and studied in connection with

these operations. The particular job on which the operation is to be made and the proper time to refer to the construction or use of the machine or tool must be decided according to conditions which obtain at a given time in a given shop.

The purpose of this text is to assist those who desire to get a knowledge of the principles and elementary operations of machine shop work. It is in no sense a treatise, nor is it a production manual. It is primarily designed to be used in connection with class talks or demonstrations in the school shop, although it may be used to supplement the information which a boy in the commercial shop may acquire by observation, practice, or other means

The author's aim has been to adapt this text for use in vocational, industrial, technical and trade schools, and in apprenticeship courses where training in machine shop practice is given. To this end he has selected what he regards as the necessary elementary information concerning machine tool operation and has endeavored to set it forth as simply and clearly as possible.

The chief function of the school shop is to teach boys to operate machines *intelligently*. It must be conceded that a knowledge of the construction of the machine tools and of the principles underlying their operation makes the essential difference between a machine operator and a machinist. Therefore, the mechanisms of typical standard machine tools have been described more or less in detail. Such descriptions should increase the interest that the boy will have in his shop work.

Many of the questions which are incorporated in various chapters of this book were originally printed in the Manual for Machinists, which was prepared by the author of this text for the War Department, Committee on Education and Special Training.

The author is sincerely grateful to the many manufacturers who have assisted him in the preparation of this book. Further, as a teacher, he desires at this time to acknowledge the help which the manufacturers render to teachers and students

by generously sending catalogues, pamphlets, and instruction books for school use. In conclusion, he wishes to express his appreciation and thanks to Mr. Frank E. Mathewson, Director, Technical and Industrial Department, Wm. L. Dickinson High School, for constant encouragement, and to his fellow teachers, Mr. Carlos H. Handforth, Mr. Henry Ouram, Mr. Paul F. Weld, and Mr. George C. Witt for honest criticism and valuable suggestions.

H. D. Burghardt.

Jersey City, N. J.

TO THE STUDENT

This is written for the young man in the shop who hopes to be a machinist. It is written for the ambitious young man (and most young men are ambitious) who expects to be, some day, a first-class machinist, a foreman, a superintendent. It is written for the alert young man, the one who combines with his ambition a determination to work, to study, to think—a determination to make enthusiasm in his work overcome laziness, to make preparedness bring the opportunity. This shop you are in is one of a hundred, or of a thousand, or of ten thousand, where certain boys are being trained. Some boys are lazy and shiftless; they will be drudges all their lives. Others are the future foremen, superintendents, managers, owners. Are you "on the job"? Who is happier, the wide-awake fellow with his feeling of satisfaction in accomplishment, or the lazy fellow with his feeling of envy? Both have to work! What is work? Webster says, "Work is a physical or intellectual effort directed to some end." Either one, physical effort or intellectual effort, taken alone is drudgery. Properly combined they produce enthusiasm. Football is work, it is sport too; but beef alone or brains alone never made a football player. Machine shop practice is work. Often and especially at the beginning, it may be hard work and dirty work, but combined with the proper amount of "intellectual effort" it is increasingly interesting as one progresses.

What is meant by this training—this doing and this thinking—in machine shop work? Where does it begin and where does it end? It means a development of the hands, ears, eyes, and mind in the power to do, to listen, to observe, to remember, and to reason.

Machine shop practice consists of certain mechanical principles that are a part of all machine shop work everywhere—

the principles of cutting tools, cutting speeds and feeds, actions of gears, screws, cams, etc.—applied in the construction of certain machines and tools and in the various machine operations: that is, in the methods of holding and doing the work. There are only a few principles, comparatively, but there is no end of methods. Machine shop training begins with the elementary principles, the easiest mathematical problems, and the simplest methods of applying these principles and problems on some kind of machine. It advances step by step, to other principles and to the application of all these principles in the doing of work, by various methods, on the other machines.

This text has been prepared with the single purpose of helping the boys in machine shops to gain quickly a working knowledge of the principles of machine work and their application in shop practice. After all, no text, no teacher, no foreman, or no friend can help the boy who is not willing to help himself.

THE AUTHOR.

CONTENTS

THE LATHE

CHAPTER I

THE MACHINIST'S TRADE

CHAPTER II

LATHE CONSTRUCTION AND MANIPULATION

CHAPTER III

Cutting Tools and Cutting Speeds

CHAPTER VIII

Chucking Work

CHAPTER IX

Tapers and Angles

CHAPTER X

Threads and Thread Cutting

CHAPTER XI

Faceplate Work

BENCH WORK

CHAPTER XII

Hammers, Screw Drivers, Wrenches, Hack Saws

CHAPTER XIII

Laying Out

CHAPTER XIV

Chipping, Filing, Scraping

FORGE WORK

CHAPTER XV

Soldering, Brazing, and Babbitting

CHAPTER XVI

Hardening and Tempering Steel

CHAPTER XVII

Hand Forging in a Machine Shop

APPENDIX

MACHINE TOOL OPERATION

THE LATHE

CHAPTER I

THE MACHINIST'S TRADE

This has been called the mechanical age. Machinery is everywhere. Practically all of the necessities and the luxuries of life are made by machinery.

In thousands of factories foodstuffs are refined and prepared, fabrics are woven, and wood and metal are fitted to make the furnishings of civilization. Everywhere factories are building labor-saving devices for the homes and workshops and building other devices for the education, recreation, and prosperity of the people. But, without the machinist there could be no engines or dynamos to furnish the power; there could be no machines because the machinist is the producer of them all. As a matter of fact there would be no factories.

Lincoln had no telephone in the White House, or no electric light. Think of the development of the telephone and the machines that have made it possible. Think of the development of the electric light and of the huge engines and dynamos in the electric power stations. Think of the growth of the automobile industry, of the improvements in the product year by year, and the improvements in the methods of manufacture. Consider the motor alone—the absolute reliability of the materials used, the necessary perfection of fit of its component parts, the marvelous methods of manufacture to make these parts by the thousands. These examples could be multiplied indefinitely—sewing machines, cameras, radios, moving-picture machines, typewriters, cash registers are a few that are universally known. Everybody knows too that all of these things are made by machinery. Who invents the machines? Who develops the machines? Who builds the

machines for making all these things? The loafer? No. The worker and thinker? Yes.

A few years ago there were two kinds of steel, "machine steel" and "tool steel." Today there are dozens of special steels—steels for gears, for shafts, for screws, for springs, for tools, even a kind of steel for the particular gear or axle or tool. The new methods of heat treatment of steel have added strength, toughness, and temper far beyond the dreams of the past generation.

Since 1927 there has been developed, to an amazing degree, the use of the so-called carbide cutting tools. Work that no one thought possible to machine, outside of grinding, is now drilled, reamed, turned, milled, and planed with cemented-carbide-tipped tools, and cutting speeds so fast as to seem incredible are recommended.

To take care of the increased power and speed needed, in all kinds of machine tools, such developments as scientifically designed main castings; ball bearings and roller bearings; heat-treated, ground, and lapped gears; centralized oiling under pressure; individual motor drive for machines and even for machine units; multiple-disk clutches and powerful brakes are demanded in modern production plants. And keeping pace with other improvements are safety appliances and guards.

These typical industrial developments together with thousands of others have been made possible by the cooperation of trained men—machinists, designers, electricians, chemists, and metallurgists working in harmony with progressive businessmen.

Progress means the development of improvements and the machine is the instrument of progress.

The machinist's trade is a great trade—great in its vital necessity, great in its ever-increasing interest, and great in the opportunity it offers for advancement. John Fritz, who founded the Bethlehem Steel Company, was a machinist in his youth; George Westinghouse started his lifework in a machine shop. Dr. John A. Brashear, who was one of the greatest

telescopic-instrument makers and one of the greatest scientists America has produced, was once a machinist. Henry Ford's knowledge of machine-shop work has served to make happy tens of millions of people.

In the light of the advance in this marvelous machine era, the machinist's trade would seem to be very difficult but the fact is, it comprises today, as it did 50 years ago, the mastery of only a few fundamentals. To be sure there are a thousand different kinds of machines to be built and assembled, but the machine-shop principles remain the same.

The operator of a special machine for doing a certain class of work, whether in a machine shop or in a factory, is not a machinist; he is only a "machine operator." Such work usually calls for very little knowledge and less judgment. It is deadly monotonous and offers no particular chance for advancement.

It is the young man who is not satisfied with being a machine operator or a "machine hand" who is determined to be something more than "cheap help"; it is the wide-awake, thinking young fellow who becomes the expert journeyman mechanic, then the foreman, and finally the superintendent. It is not all fun; it means work; it means study; it may mean sacrifice of money at the start, but it is interesting, vitally stimulating, and not deadening—walking a treadmill is only existing; going ahead is living.

In the large industrial centers, more than half the workers are employed in the metal trades; that is, more mechanics and machine hands are employed in these trades than are at work in all the other trades taken together. It is safe to say that 90 per cent of the foremen in the great manufacturing plants have been promoted from the ranks of machinists, and that 90 per cent of the superintendents of these factories have previously been foremen.

It is admitted by all manufacturers that the proportion of expert machinists is growing constantly smaller; why then, should it be difficult for the young man in the shop to foresee the opportunities that may be his, *if he is prepared?*

During the last few years factory conditions have greatly changed; scientific management, improved production methods, etc., have made the machine hand more than ever before a mere part of the machine, while the standing of the real machinist is on an increasingly higher level. The same operations—turning, boring, drilling, etc.—are performed, but are made easier, more quickly, and with greater accuracy. The cutting tools have more than double the efficiency of those of a few years ago. The machines have been built stronger, more rigid, more adaptable, more accurate. While these improvements have lessened the *manual* labor incident to machine work, the truly marvelous strides made in the manufacturing of thousands of parts exactly alike have advanced the making of the special machines, tools, and gauges—that is, real machine-shop work—to a much higher plane as regards the *mental* effort necessary to do this work.

Conditions that obtain in the machine shops, obtain also, in more or less degree, in the other metal trades—in the steel mills, in the forge shops, and in the foundries. Doubtless these conditions may be found in any trade or in any business—the uninteresting, monotonous work, the drudgery, is done by unskilled or semiskilled "hands." The *trained man* has the interesting work, the most pay, and best of all, the chance for promotion.

A SHORT MACHINE-SHOP CATECHISM

1. What Is a Machine Shop?—A machine shop is a place in which metal parts are cut to the size required and put together to form mechanical units or machines, the machines so made to be used directly or indirectly in the production of the necessities and luxuries of civilization. Machine-shop work is the basis of all mechanical production.

2. What May Constitute the Equipment of a Machine Shop?—Machine-shop equipment consists in part of certain standard machine tools, the kind, the size, and the number of machines depending, of course, upon the product of the shop.

Machine-shop equipment includes, in addition, the tools used at the bench and on the floor as well as the measuring and adjusting tools, the work-holding and tool-holding accessories, and the small tools used in the machines.

3. What Are the Standard Machine Tools?—The lathe, the drill press, the shaper, the planer, the milling machine, the grinding machine, and the boring mill are usually considered standard machine tools. The turret lathe, the slotter, the

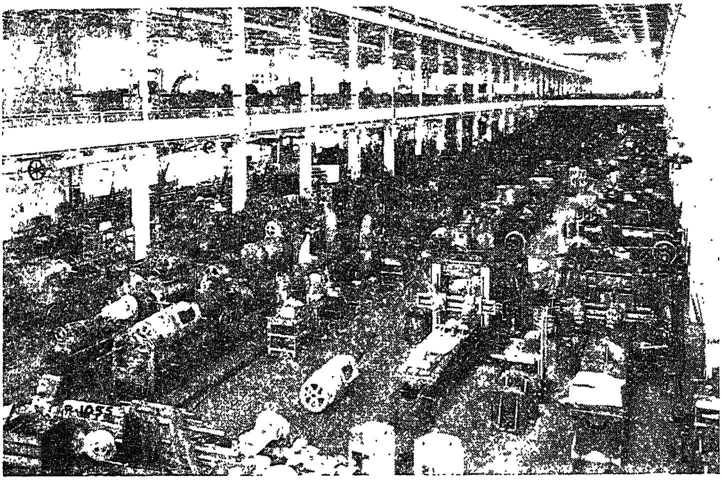


FIG. 1.—A view of a typical large machine shop. (*Courtesy of The Bullard Company.*)

gear-cutting machine, and many others are usually referred to as manufacturing machines or special machines. The propriety of the term special machine lies in the fact that these are modifications of standard machine tools and have been developed to meet specialized production problems.

It is difficult, and perhaps unnecessary, to draw the line between standard and special. A few years ago the milling machine, and still more recently the grinding machine, were considered special machines. Now they are regarded as very essential machines in the well-equipped shop. One who is able to operate intelligently the machines recognized as standard

will be able, with a minimum amount of study and experience, to understand and operate any special machine.

It must be understood that all kinds of machine tools are made in a great variety of types and sizes. Fortunately for the machinist, certain basic principles of construction and operation which obtain in one size of machine, obtain in all machines of that class, and further, many of these same principles are found in other kinds of machines. The student or

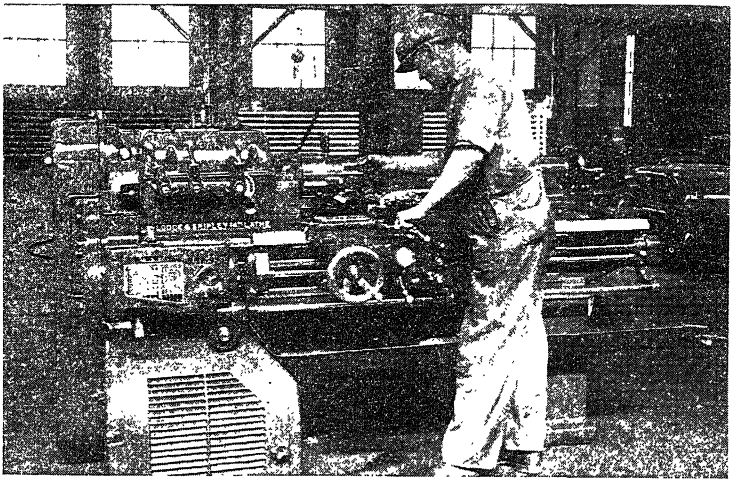


FIG. 2.—Turning in a 14-in. lathe. (*Courtesy of Lodge & Shipley Machine Tool Company.*)

apprentice does not have to begin all over again to learn to operate each size and kind of machine. If he has mastered certain principles regarding one size or type of any kind of standard machine tool he has certain knowledge that will help considerably in understanding the construction and operation of other machines. The underlying principles of the proper cutting speeds and feeds, the grinding of the cutting tools, of adjustments and measurements, apply to all of them.

The Lathe.—The lathe (Fig. 2) is a metal-turning machine tool in which the work, while revolving on a horizontal axis,

is acted upon by a cutting tool which is made to move slowly (feed) in a direction more or less parallel to the axis of the work (longitudinal feed), or in a direction at right angles to the axis of the work (cross feed). Either feed may be operated by hand or by power (automatically) as desired. When the feeding is in a direction parallel to the axis of the work, cylindrical or "straight turning" is accomplished. When the

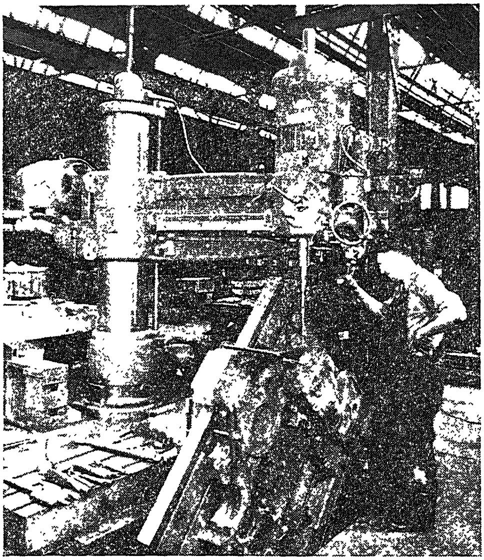


Fig. 3.—A special job in a 6-ft. radial drill. *(Courtesy of The American Tool Works Company.)*

cut is in a direction at a slight angle to the axis of the work a "taper" is the result; more of an angle results in "turning to an angle." The cut at right angles to the axis of the work (the cross-feed operation) is known as "facing" or "squaring." Cutting inside of a hole is termed "boring."

The Drilling Machine.—The drilling machine or "drill press" is a machine tool used mainly for producing holes in metal. In this machine the work is securely held while a revolving cutting tool is fed into it. The cutting tool most commonly used is called a "drill"; it has, in effect, an action

similar to a wood "bit." In Fig. 3 a medium size radial drill is illustrated.

The Shaper.—The shaper (Fig. 4) is ordinarily used for finishing flat or partly curved surfaces of metal pieces few in number and not usually over a foot or two long. In the shaper, the cutting tool has a reciprocating (forward and return) motion, and cuts on the forward stroke only. The work is usually held in a vise bolted to the work table and the

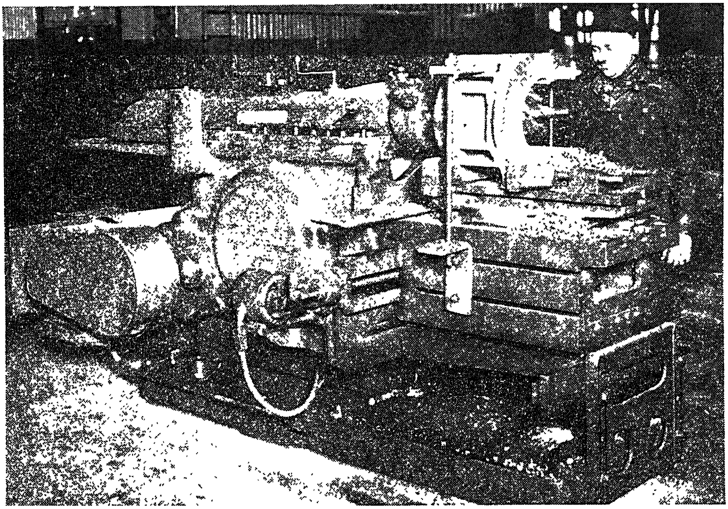


FIG. 4.—A special job in a 24-in. shaper. (*Courtesy of The American Tool Works Company.*)

regular feed is accomplished by causing the work table to move automatically at right angles to the direction of the cutting tool. The construction of the tool head permits of down feed at right angles to the regular feed, or at any other angle if desired. The cutting tools used in the shaper are similar to the turning tools used in the lathe.

The Planer.—The planer (Fig. 5) is a machine tool used in the production of flat surfaces on pieces too large or too heavy or perhaps too awkward to hold in a shaper. In this machine the table or "platen" on which the work is securely fastened

has a reciprocating (forward and return) motion. The tool head may be automatically fed horizontally in either direction along the heavily supported crossrail over the work, and automatic down feed is also provided. Cutting tools used in planer work are the same as those used in the shaper.

The Milling Machine.—The milling machine (Fig. 6) is a machine tool in which metal is removed by means of a revolv-

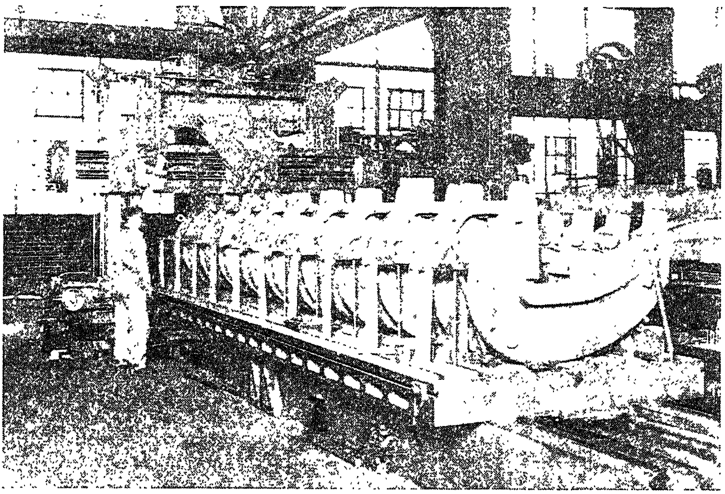


Fig. 5.—An angular cut on a "string" of punch-press bodies. (*Courtesy of Cincinnati Planer Company.*)

ing cutter with many "teeth," each tooth having a cutting edge which removes its share of the stock. The work is supported by various methods on the work table, and may be fed to the cutter either longitudinally, transversely, or vertically. A great variety of work may be done on a milling machine, and next to the lathe it is perhaps the most adaptable and interesting machine in the shop.

The Grinding Machine.—The grinding machine (Fig. 7) is a machine tool in which an abrasive wheel is used as a cutting tool to obtain a very high degree of accuracy and a smooth finish on metal parts, including soft and hardened steel. A

large variety of types and a number of sizes of surface-grinding machines and external and internal cylindrical-grinding machines are manufactured for ordinary and special grinding operations. The advance in the use of the grinding machine is one of the outstanding factors in present-day rapid, accurate production of metal parts.

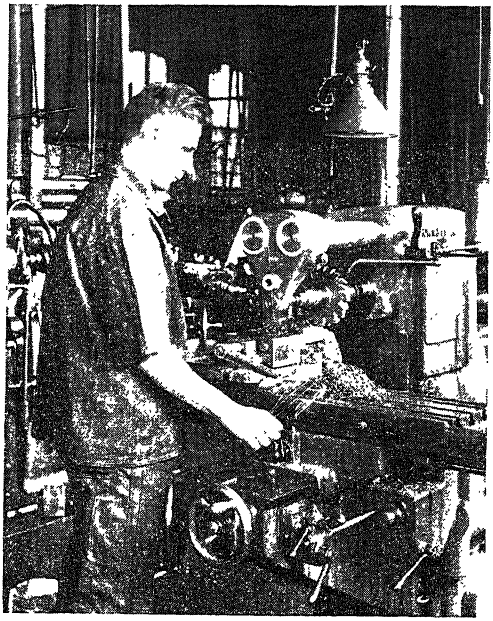


FIG. 6.—A side milling operation in a column and knee type plain milling machine. (*Courtesy of Brown & Sharpe Manufacturing Company.*)

The Boring Mill.—There are two distinct types of boring-mill design, the vertical (Fig. 8) and the horizontal (Fig. 9), both of which are modifications of the lathe. Boring is the operation of enlarging a hole, usually by means of a single cutting tool, and the boring mill is designed primarily for the purpose of finishing holes that are impracticable to finish in a lathe or other machine because of the size or shape of the casting.

In a vertical boring mill, the work table revolves on a vertical axis and the cutting tool (which may be a drill or a boring tool or a turning tool) is arranged above the table and may be fed laterally (toward or away from the center of the table) and up or down in any position. Because of these feeding arrangements, turning and facing may be accomplished as easily as boring. In the smaller sizes the various tools are arranged in a turret head.

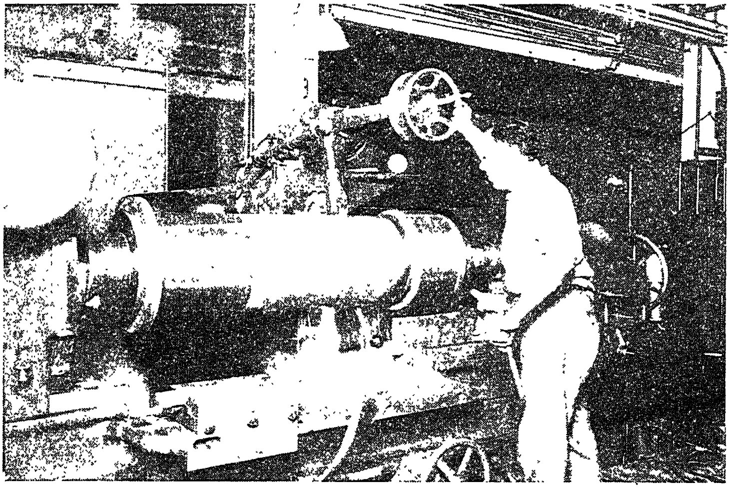


FIG. 7.—Finishing a cylindrical surface in a 28- by 120-in. hydraulic drive grinding machine. (*Courtesy of Landis Tool Company.*)

In a horizontal boring mill, the cutting tool revolves on a horizontal axis. The spindle which carries the cutting tool may be fed longitudinally through the spindle head and in the more recent designs the spindle head may be fed vertically. The work table may be fed longitudinally and transversely. The horizontal boring mill while designed primarily for boring holes may also be used for finishing horizontal and vertical flat surfaces by means of a suitable milling cutter fastened to the spindle.

NOTE.—Because of the size of the boring mills, vertical and horizontal, and the nature of their product, it is not customary for beginners in shop

practice to operate these machines. The setup of the work, the shaping and setting of the tools, the measuring, gauging, etc., in a vertical boring mill greatly resemble chuck work in a lathe. In the horizontal boring mill these operations are similar to many found in lathe and milling-machine work. That is, the mechanical intelligence necessary to operate a boring mill can be acquired by application of like principles on smaller

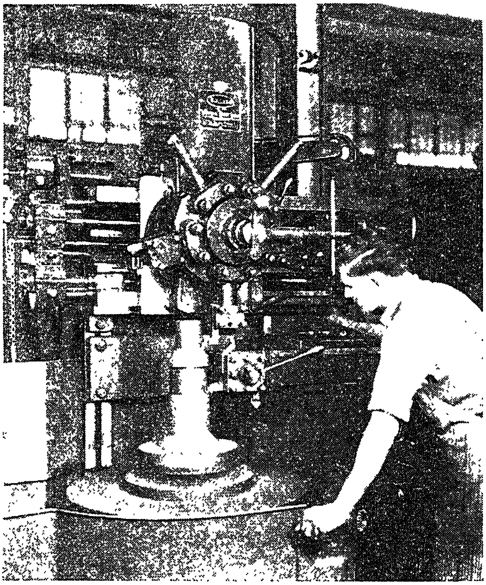


FIG. 8.—An interesting job in a vertical boring mill. (*Courtesy of The Bullard Company.*)

machines and on work less costly. Therefore, a further discussion of boring mills is nowhere included in this text.

4. What Is Meant by "Bench Work" and "Floor Work"?—Bench work in a machine shop consists of laying out, assembling, and the final fitting of parts. When the same operations are performed on heavy work, the term floor work applies.

5. Are There Many Specified Divisions in the Machinist's Trade?—There are probably more opportunities for specializing in machine-shop work than in all of the other trades taken together. Consider the range in sizes of machines and consequently the work to be done; also the opportunity of

specializing on certain types of machines, such as planer, milling machine, or grinding machine. One might prefer model work, experimental work, or toolmaking. Toolmaking itself is divided into several branches such as diemaking, jig making, gauge making, etc. In any event, the machinist of whatever class, or the toolmaker of whatever specialty, must have a certain *machine-shop sense* and a knowledge of

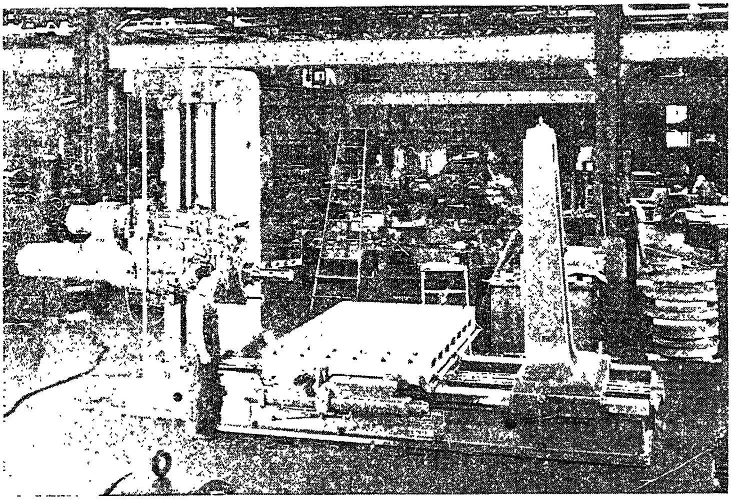


FIG. 9.—A horizontal boring mill. (*Courtesy of William Sellers and Company.*)

principles and methods. These can be acquired only by experience and study.

6. How Are the Employees of a Commercial Manufacturing Shop Classified?—The machine shop is but one of the essential units in the typical production plant. In the machine shop the special tools and machines are developed and built, and repairs made. Elsewhere in the factory are rooms filled with special machines or "manufacturing" machines run by machine operators. These operators become very skillful in doing one thing, but they are not mechanically trained.

It has not seemed necessary or advisable for manufacturing purposes to train but a small proportion of the employees to be

anything but operators, or assemblers, or machine hands, with the result that few, indeed, are what may correctly be called machinists.

The employees of commercial manufacturing shops may be classified according to grades of attainment about as follows:

Machine Operator.—A machine operator is one who merely operates a "manufacturing" machine doing one class of work. He is able to start and stop the machine, fasten in place the piece to be machined, and remove it when the operation is complete. He makes no cutting-tool adjustments and is in no sense a mechanic.

Assembler.—The assembler takes the parts already made and inspected and puts them together. In general, this work calls for some skill and a reasonable amount of common sense, but requires no particular mechanical intelligence except where the final fitting and adjusting is done on the job.

Machine Hand.—A specialized machine hand is one who has very little general machine-shop knowledge but who has operated a special machine long enough to be skillful in a variety of work on this machine, or on a machine of this class. He is able to do his own setup work and make the necessary adjustments.

Machinist's Helper.—A machinist's helper knows the names and uses of the various small tools (cutting tools, measuring tools and gauges, holding tools, etc.) used in machine-shop work. In addition he may be able to do elementary bench work or machine work.

A Specialized Machinist.—A specialized machinist is one who has had some general machine-shop experience and has made a specialty of some one machine or some one class of work, such as lathe work and planer work. He has a broader background of experience and more versatility than the machine hand.

Bench and Floor Hands.—Bench hands and floor hands possess information and skill regarding a number of so-called hand operations (as differentiated from machine operations) such as filing, scraping, assembling, and adjusting. A skilled

bench hand or floor hand, in general machine-shop work, has the ability also to read blueprints readily and to do layout work. In addition, a first-class bench hand or floor hand has had, usually, considerable experience in machine operation.

The Machinist.—The general machinist has had enough experience, has acquired enough information, has developed enough judgment, and possesses "head" enough to be able to

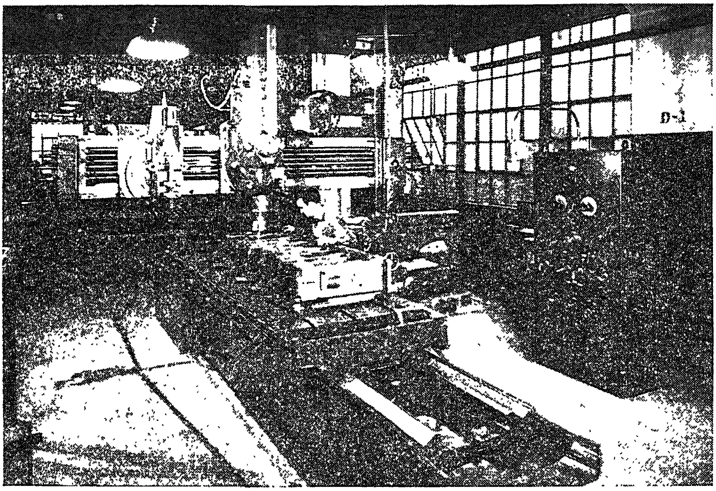


FIG. 10.—A job on a planer-milling machine for a first-class machinist. Surfaces must be exactly planed and milled, with holes and slots finished to size required. (*Courtesy of The G. A. Gray Company.*)

set up intelligently and operate any standard machine tool and perform any bench or floor operation. In addition, he is able to harden and temper machine-shop cutting tools.

The Toolmaker.—The expert toolmaker qualifies substantially as the general machinist. Toolmaking is usually a lighter or smaller class of work and, generally speaking, involves more delicate workmanship, more accurate measurement than does general machine work. It also involves more mathematical calculations on the part of the workman and a more extended use of the various machine-tool attachments.

Apprentice Machinist.—The grades of apprentice vary naturally from beginning apprentice to advanced apprentice. The beginner may have no previous machine-shop experience, while the advanced apprentice should have a thorough training in the fundamental knowledge of a machinist. The typical apprentice agreement is based upon an understanding that the apprentice shall be given a few months' experience on bench and floor work and on each of the standard machine tools, and, in addition, shall be given an opportunity to learn the essential principles of the operation of each. The degree of attainment of an apprentice at a given time depends of course upon the individual, other things being equal. The ambitious apprentice is keenly desirous of learning the trade. Where the employer fully meets his obligation, the apprentice is offered every advantage to learn the operations, methods, calculations, and principles involved in machine-shop practice, to the end that his development may be rapid and sure.

7. What Is the Knowledge One Must Have to Be an Expert Machinist?—He must have an understanding of certain fixed principles which obtain in all machine-shop practice, for example:

The action of metal-cutting tools.

Cutting speeds.

Feeds and feeding devices.

Strength of materials—stresses and strains—rigidity and spring.

Gear trains.

Measurements.

Adjustments, etc.

He must have a sufficient knowledge of arithmetic to read measurements from the various instruments, and to make the necessary calculations for cutting speeds, gear velocities, angles, threads, etc.

He should have a sufficient knowledge of the principles of mechanical drawing to be able at least to read blueprints of machine details.

He should have a reasonable working acquaintance with the construction and operation of the typical standard machine tools. To be an expert machinist does not imply a highly specialized knowledge of all or perhaps of any one of the machine tools, but it is an established fact that the high-class

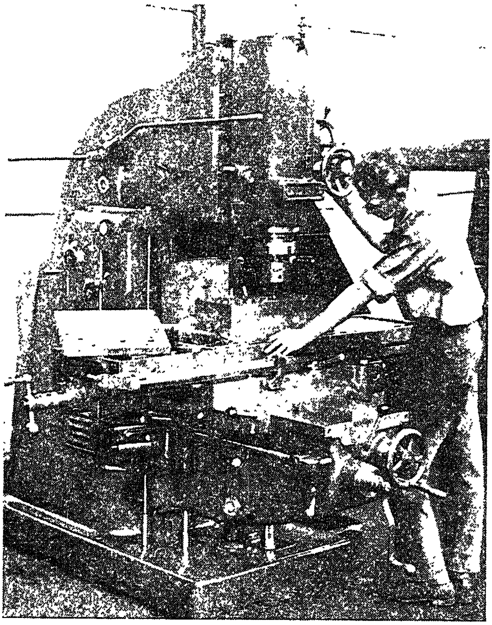


FIG. 11.—A toolmaker's job in a vertical milling machine. Boring a series of holes that must be accurately sized and exactly spaced. (*Courtesy of Cincinnati Milling Machine Company.*)

specialist on a particular kind of machine or class of work is also to a considerable extent familiar with general machine-shop practice.

He should be well acquainted with the characteristics of the metals used in machine construction, particularly cast iron, bronze, aluminum, and the various kinds of steels. And he must be familiar with the heat treatments of carbon steel and high-speed steel.

He must be resourceful in methods. The most efficient way of doing a certain job often depends on the accuracy required; the number of pieces to be made; the available machines; and the available tools.

He must have a considerable knowledge of the sequence of operations; the knowledge of how to go at the job to assure accuracy of result in the shortest time. It is said that there are over 100 operations on the receiver of the Springfield rifle, which, when finished, weighs about a pound. Think of the satisfaction of being able to arrange the sequence of operations and to design the special tools and fixtures for such a job. It requires the knowledge of a machinist. Every job of five operations or of ten operations thoughtfully worked out is a problem solved and every problem solved is a help in solving the next.

8. What Chance Has a Machinist for Promotion?—The chief advantage of the machinist's trade is in the opportunities it offers for promotion. Every machine-shop foreman, naturally, must have been promoted from the ranks. Further, practically every superintendent and every master mechanic of any industry manufacturing metal goods of any description, is a machinist. These men may have gone through the drawing room, but they were machinists before they were draftsmen.

Thousands of successful manufacturers were once machinists. They had ideas suggested by their machine-shop experience. They put these ideas into practice and developed them. To mention only a few: Joseph R. Brown and Lucien Sharpe, founders of the Brown & Sharpe Manufacturing Company, of Providence, Rhode Island; Francis A. Pratt and Amos Whitney, founders of the Pratt & Whitney Company of Hartford, Connecticut; Worcester R. Warner and Ambrose Swasey, founders of Warner & Swasey of Cleveland, Ohio, were all apprentice boys, and then machinists, before they were counted among the foremost manufacturers of the world.

9. What Are the Essential Characteristics of a Machinist?—Carefulness, orderliness, accuracy, speed, judgment, and con-

fidence are six essential characteristics that a skilled machinist must have.

Care of Self.—A machine is a good servant but a cruel teacher. It is a dreadful thing to lose a finger in order to learn that revolving gears are dangerous things to handle. It is better to be overcautious until habits are formed, which, without conscious reasoning on the part of the operator, make a dangerous move around a machine practically impossible. A well-trained machinist is careful through habit.

Care of Machine.—A mechanic is always careful not only of the appearance but of the good condition of his machine.

Orderliness.—The truth of the need of "a place for everything and everything in its place" is nowhere better exemplified than in a shop where a number of people at various times use the same machines and tools. And orderliness makes one's own work easier and speedier. Orderliness and neatness about the machine and bench are the marks of a good workman. They are habits worth forming.

Accuracy.—It is very often necessary for a machinist to work within $1/1000$ in. This is easy enough with the machine tools and measuring tools found in modern shop equipment. It means, however, that the machine must be perfectly adjusted and otherwise in first-class condition, that the cutting tool must be properly sharpened and set, and that the measuring tool is dependable for accuracy.

Speed.—An expert mechanic studies the methods and means of doing a job; makes sure that the machine, cutting tools, and measuring tools are in good condition, and then with care, and without undue haste, operates the machine to obtain the maximum production. Carefulness, orderliness, thoughtfulness, and close attention to the little things make for speed.

Judgment.—A man is successful in any business in about the same proportion as he acquires judgment. This is as true of a machinist, foreman, or superintendent, as it is of any other business or professional man.

Judgment is the ability to decide correctly after comparing ideas, methods, or facts. The mechanic must cultivate ideas,

study methods, and learn facts regarding his trade. He must know when to rough, when to finish, where accuracy is necessary, and when and where it is not essential. He must be resourceful in ideas and methods in order to adapt himself to various shop conditions. Judgment is intelligence, and every job, well thought out and well done, sharpens the intellect and paves the way toward success.

Confidence.—The man who through study, thought, and careful application has confidence in his own ability to accomplish results has in this confidence a factor which makes for success.

10. How Are These Characteristics Acquired?—A skilled artisan is one who has the power to think and execute with knowledge and ability. To think is to employ the mental capacity of distinguishing ideas and methods. To execute with expert ability means to employ the senses with confidence and accuracy.

The man who aspires to leadership in any trade or profession must *study* and must *work*. Theory and practice walk hand in hand toward skill, knowledge, and efficiency, and a man's value to himself and to his employer is always in proportion to his efficiency.

Skill in machine work may be acquired by studying how and why certain operations are done and in connection with this study, a considerable experience in performing these or similar operations is essential.

A lifetime of merely doing is not sufficient to acquire knowledge, except in a very limited degree. One must take advantage of what others have done and are doing. A fund of information is available in the special articles in the trade papers, and in the advertisements in these magazines; in the manufacturer's catalogues and instruction bulletins; and in reference books of which dozens have been prepared regarding each of the standard machine tools. Whole volumes have been written also about machine parts such as gears and cams; and about machine-shop mechanics and machine-shop mathematics; about steel and the heat treatment of steel

for various purposes. It is unnecessary to own all of these books but it surely is advisable to know where certain kinds of information can be found when wanted. It is almost as necessary for a machinist to appreciate the value of a reliable handbook as it is for him to know how to use a micrometer. To keep up to date it is well worth while to read regularly at least one of the magazines relating to the work of the machinist. The progressive machinist is a student.

Efficiency can be approached only through the application of the best methods. The selection of the best method requires sound reasoning. The power of sound reasoning is founded in the knowledge of principles. One must know *why* before he can reasonably know *how*.

This sounds serious; it is serious, but certainly not discouraging. Study is easy and work is fun when one is interested. Master the first principles and get interested, develop that interest into the right kind of enthusiasm, and your knowledge and your power, your good influence and your income, will grow and grow fast.

CHAPTER II

LATHE CONSTRUCTION AND MANIPULATION

11. Introduction.—In this chapter it seems advisable to discuss the simpler kinds of lathe units and drives, even what the present-day salesman might call the obsolete gear arrangements for speeds, feeds, and leads, as well as some of the newer designs. There are three reasons for this: (1) The same *principles* are involved in the later models as are found in the older cone-pulley and back-gear drive, as for example in the gear ratios for spindle speeds, feeds, and cutting screw threads; (2) there are many of the older machines in most shops, and these must be operated and should be understood by beginners; (3) a knowledge of the older, simpler construction is a decided advantage in understanding the newer, more intricate designs.

The line between the old and the new, if it could be drawn, is of no particular importance. The fact is, the new gradually develops from the old, and the parts of this chapter are arranged accordingly.

12. The Engine Lathe.—The lathe while being the most important machine-shop tool is also one of the simplest in its construction. Its simplicity, together with the wide range of operations of which it is capable, makes it especially interesting to the young mechanic. This, in addition to the fact that so many basic elements of machine construction and machine-shop practice are involved in the construction and operation of the lathe, makes lathe work without question the proper elementary machine-shop work.

The function of a lathe is the removal of metal, by means of a suitably formed cutting tool of hardened and tempered steel, from a piece of work which is securely supported and made to revolve.

The engine lathe or "lathe" as it is usually called in machine shops is power driven, has automatic feeds, and is provided with a lead screw for cutting threads. These machines are classified as to their size by the maximum diameter of work which may be revolved over the ways, such as 10, 14, 16, 24 in., etc. For particular classifications the total length between centers is also noted. The smaller lathes, 14 and 16 in.

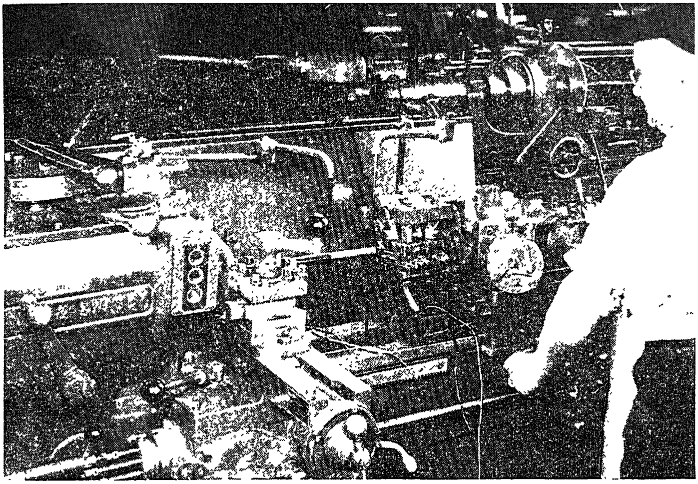


FIG. 12.—Several operations, in one setup, in a turret lathe. (*Courtesy of The Warner & Swazey Company, Cleveland, Ohio.*)

(Fig. 2), are far more numerous than the larger sizes; lathes with 36-in. swing and larger are, however, very necessary in manufacturing heavy ordnance, huge pumps, engines, shafts and rolls.

The general lathe operations are straight (cylindrical) turning, taper turning, boring (straight or taper), facing (which is a cut at right angles to the axis of the work), and thread cutting. Attachments of particular value are available for special operations, such as the relieving or backing-off attachment, grinding attachment, and taper attachment.

Special lathes of a large variety of patterns and sizes are made for different kinds of work, the most notable example

of which is the turret lathe (Fig. 12). The turret lathe is a "manufacturing" machine. Considerable mechanical ability is required to make and adjust the several cutting tools in the turret head and cross slide, but when the tools are once made and adjusted and the stops set, it requires no particular mechanical intelligence to operate the machine. To run an engine lathe on a variety of work is much more interesting and calls for a higher degree of intelligence than the operation of a turret lathe.

13. Running a Lathe.—Almost every young man in the shop—errand boy, apprentice boy, machinist's helper—wants to "run a lathe." It is a worthy ambition, but before he can hope to do much more than start or stop the machine, he must learn about the cutting tools—their shape, how they are sharpened, and how they are held to peel off the metal. He ought to know how to read the thirty-seconds and sixty-fourths of an inch on a rule or "scale" quickly and accurately, how to "feel" with a caliper to obtain a measurement within 0.002 in. He should learn as soon as possible the names and functions of the parts of the machine. He should appreciate, to a reasonable extent, the value of the proper cutting speeds and feeds. He should know how to oil the lathe carefully and thoroughly. After he has studied these things "running the lathe" will be more interesting.

It is not to be expected that the beginner will learn all about the construction of a lathe in one or two lessons. This chapter covers information that should be acquired as rapidly as possible but may be acquired in connection with the doing of jobs in the lathe. The right kind of boy will not be satisfied with merely operating the lathe any more than he will be satisfied with sitting down to study names and functions of parts, and theory of why and how. *Experience* and *knowledge* go together and make for keener interest and faster progress.

The real mechanic understands the construction of his machine; he knows the names and uses of the parts and the principles underlying the operation of the mechanisms. The

more one learns of these things the more interesting the work becomes.

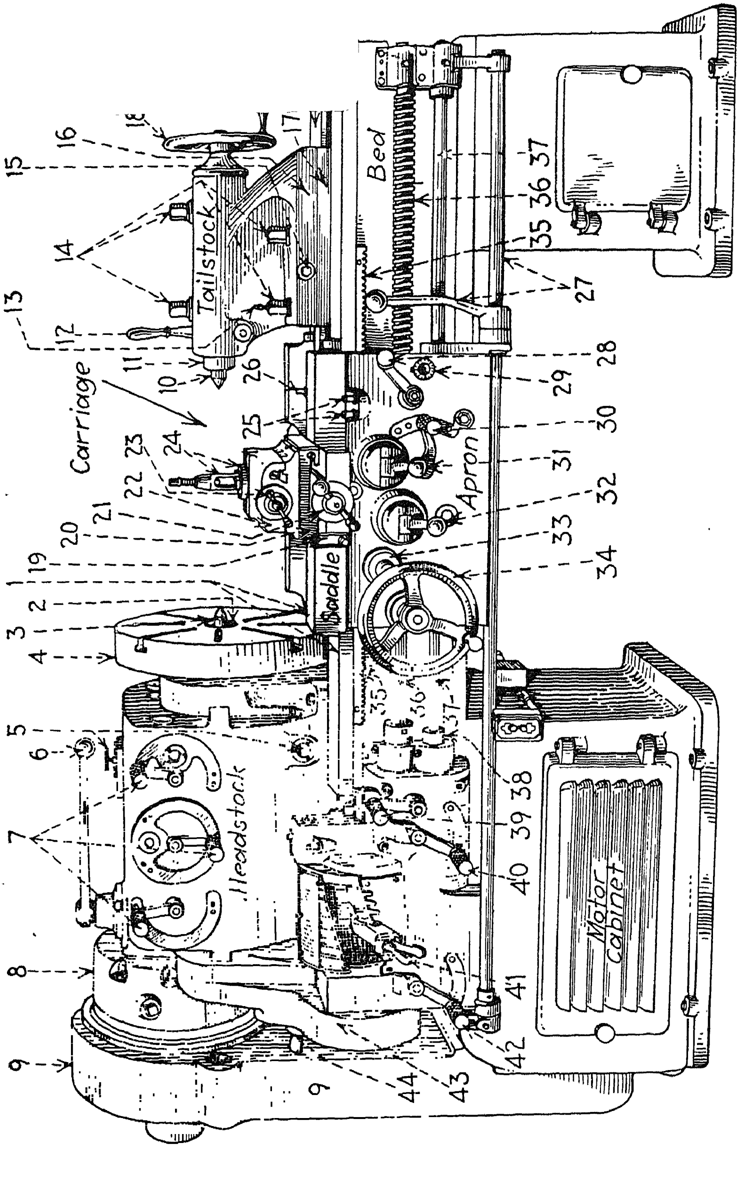
14. Parts of the Lathe.—On the following pages is illustrated a standard engine lathe with the parts numbered and named. Do not be satisfied until you know the name and function of each part.

15. Cleaning and Oiling.—One of the best ways for the beginner to start his acquaintance with a machine is to clean it thoroughly and oil it. Cleaning should be done while the machine is idle; *never when it is running.* A small piece of waste moistened with kerosene will serve to cut the dirt and grease, after which wipe with dry waste. The ways and other exposed bearings should be especially clean before oiling. Use a stick to get in the corners, make a good job of it.

Oil the ways and other flat bearings (the dovetail bearing of the cross slide and over back of the bed where the carriage gib slides) by rubbing on the oil with the fingers.

Every piece that revolves has one or more bearings and every bearing has to have oil. Find every revolving part by turning the lathe by hand. Find the particular oil holes, be sure they are not stopped up with dirt, and put in sufficient oil to lubricate the bearings thoroughly. Common sense will help one to judge how often a bearing should be oiled and how much oil to use. It is perhaps sufficient to say that it is a crowning disgrace to let a machine get "stuck," and also that a bearing flooded until the oil drips on the floor is an indication of ignorance or carelessness.

Some of the later models of machine tools have pressure lubrication. A pump forces the oil from a reservoir through a filter and then through pipe lines to the bearings and gears. A gear splash, a cascade, or a pipe direct to the bearing may be used. Oil grooves are scientifically laid out in round and flat bearing surfaces. A strainer or settling basin is used to separate foreign matter, and a filter further to clean the oil is provided. *Having all these improvements in no wise lessens the responsibility of the machinist in being sure that his machine is properly oiled.*


Headstock
Tailstock
Carriage
Saddle
Apron
Bed
Motor Cabinet

While the student is cleaning and oiling is an excellent time to learn more concerning the features of the machine. Short descriptions of the unit parts of the lathe are given immediately after the following hints.

16. A Few Suggestions:

1. Roll up your sleeves.
2. Do not wear a ring.
3. Keep the wrenches, measuring tools, etc., arranged, not thrown, on the lathe board. Never put work, files, tools, etc., on the ways of the lathe, but do put a little oil on them occasionally.
4. Then put the oilcan where you can't jab your face against the spout.
5. Keep your hands away from revolving gears.
6. If you prefer to clean with your finger, the hole in the spindle or a hole you are boring, *stop the machine* or you may leave your finger in the hole.
7. Ask questions after a reasonable amount of thought and study.
8. Do not move a handle to see what will happen, especially if the machine is running; reason out what the handle is for, or learn in some way, and then move it. Make sure you are right before starting the machine.
9. If you make a mistake do not make the second one of trying to cover it up. The first may sometimes be excusable, the second never. Remember everyone respects an honest straightforward chap.
10. Do not make the same mistake twice.

DESCRIPTIONS OF LATHE UNITS

The engine lathe may be said to comprise six essential features: the bed, the headstock, the tailstock, the carriage, the feeding mechanism, and the thread-cutting mechanism. Look up the first four in Fig. 13 and also find the *feed rod* (37) and the *lead screw* (36). It is the feed rod that transmits the feed motion to the carriage at whatever position on the bed

the carriage may be, and the lead screw serves to move the carriage when screw threads are being cut. Their value will soon be understood.

17. The Bed.—The bed is of sufficient depth and width to give rigidity under heavy cuts and is braced inside by cross girths to give stability and strength. Ways or V's are machined and scraped[1] on top of the bed. The outside or carriage ways afford a perfectly aligned track for the travel of the carriage. The inside ways furnish a permanent seat for the headstock, and a perfectly aligned seat for any desired position of the tailstock. These ways are usually about 90 deg. included angle, and have the tops well rounded to prevent bruising.

18. The Headstock.—The headstock complete comprises the headstock casting; the main spindle; the necessary mechanism for obtaining the various spindle speeds; and also certain gears which are used in transmitting motion from the spindle to the feed rod of the feeding mechanism, and to the lead screw of the thread-cutting mechanism. The *main spindle* of the lathe revolves in two bearings, one at each end of the headstock. These bearings are very accurately made and assembled to bring the axis of rotation of the spindle parallel to the ways, that is, "in alignment." A hole extends through the entire length of the main spindle and this hole is bored taper at the front end to receive the live center. The bearing surfaces of the spindle, the front end, or "nose," and the taper hole for the live center, are made mechanically accurate and true and great care should be exercised to keep them so. The main spindle of a machine controls the speed of the work, when the work revolves, as in a lathe; or the speed of the cutting tool, if the tool revolves as in a drilling machine. It must be mechanically true and perfectly aligned because a spindle out of true in any way, or in imperfect alignment, or with improperly adjusted bearings, will cause trouble.

19. The Tailstock.—The tailstock comprises the tailstock spindle; the tailstock slide in which is bored the housing for

[1] Scraping gives cast iron its best bearing surface (see page 318).

the spindle; the base which is fitted to the inside ways; the screw and handwheel which control the movement of the spindle; the device for clamping the spindle; and the tailstock clamping bolts. The tailstock is for the purpose primarily of giving an outer bearing and support for work being turned on centers. To accommodate different lengths of work, it may be moved along the inside ways and clamped in any position, and in addition, the tailstock spindle (11) (Fig. 14), which holds the dead center (10), is adjustable longitudinally by means of a handwheel, which operates the screw (*S*). The

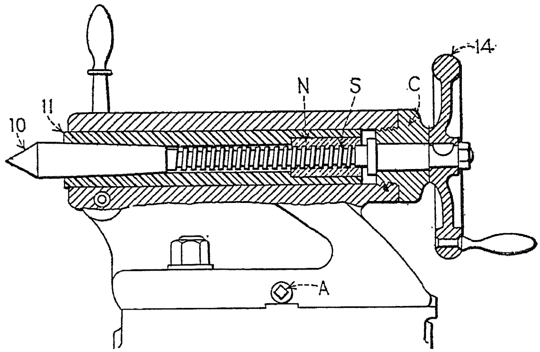


Fig. 14.—Partial section view of tailstock.

tail spindle is carefully fitted in its housing and is normally in exact alignment with the main spindle. There is no *vertical* adjustment whatever, but the tailstock slide may be adjusted *transversely*, that is, towards or away from the operator, by means of the adjusting screws *A* (a corresponding screw is on the other side of the tailstock). A keyway (or spline) is cut about two-thirds the length of the spindle from the inside and a key located on the back side of the housing serves to keep the spindle from turning while allowing it to slide freely. The spindle is hollow. It is bored taper on one end to receive the center, and counterbored on the inside end to receive and hold the bronze nut *N*. The shoulder screw *S* may revolve freely in the cap *C* by turning the handwheel (14) but has no

end motion. Hence as it turns in the nut N it causes the spindle to move towards or away from the handwheel depending upon which way the handwheel is turned.

To Remove the Dead Center.—Turn the handwheel "back" until the end of the screw hits the end of the center and forces it out.

Cautions.—If the spindle is turned out so far as to run off the screw, be very careful to see that the keyway lines up with the key before turning it back.

Be careful not to turn back too far or the spindle will jam against the shoulder of the screw.

20. The Carriage.—The carriage consists of the saddle and the apron. The *saddle* is fitted to the outside ways and is gibbed to the bed. It is in the form of a letter H, being bridged across the lathe bed to carry the cross slide and tool rest. The particular function of the carriage is to carry the cutting tool. It may be caused to move ("feed") by hand or by power along the outside ways, lengthwise of the bed or "longitudinally," and further, the crosspiece of the saddle is machined to provide a way for the tool-rest slide so that the cutting tool may be moved by hand or by power to give a "cross feed" at right angles to the "long feed." The *apron* contains the gears and clutches for transmitting motion from the feed rod to the carriage, and also contains the split nut (or half nuts) which engages with the lead screw when cutting threads.

21. The Feeding Mechanism.—Two or more gear trains (series of gears in mesh) one of which is located within the headstock (the "reverse gears") and the others at the end of the lathe ("ratio gears" and "change gears") serve to transmit motion from the main spindle to the feed rod which extends along the entire length of the bed. From the revolving feed rod, motion is transmitted through various gears which are located in the apron to cause the carriage to move on the ways, called *longitudinal feed;* or when *cross feed* is desired, to cause the cross slide to move transversely. The motion of the feed gears in the apron is controlled by means of frictions, and the controls for both longitudinal and cross feeds are

located on the front of the apron within easy reach of the operator. (For friction clutch see footnote, page 56.)

It sounds like a lot of gears but really there are only a few, and the way they work is quite simple and very interesting, as will be found a little later.

The object of having the long feed rod is to be able to transmit feed motion at any position of the carriage on the bed. Notice the small gear, fixed in its position in the apron, but sliding along the feed rod as the carriage moves. It acts equally well in any position on the feed rod.

22. The Thread-cutting Mechanism.—The thread-cutting mechanism includes the necessary gears to transmit motion from the main spindle to the lead screw. These may be, and usually are, the same gears used to transmit motion to the feed rod. The lead screw extends along the bed above the feed rod. It is of substantial diameter with a fairly coarse thread cut with great accuracy.

Motion of the lead screw may be transmitted to the carriage by closing the two halves of a split nut over the screw, the split nut being securely fastened to the apron. *The thread-cutting mechanism should not be used for "feeding" and the feeding mechanism cannot be used for cutting threads.*

NOTE.—The feeding mechanism and the thread-cutting mechanism are more fully explained, beginning page 49.

23. A Very Important Precaution.—A habit one should cultivate when learning to run a lathe is to make sure that the carriage moves freely on the ways before starting the machine.

The first thing an experienced machinist always does when going to work on a lathe is to move the carriage on the ways by the hand feed to make sure

1. That the split nut is not tightened.
2. That the feed control is not tightened.
3. That the carriage clamp screw is not tightened.
4. That the ways are oiled.

If the split nut and feed are both tight when the lathe is started the apron will be broken causing several dollars damage. If either is put in when the carriage clamp screw is tightened, the apron mechanism will be strained and injured. If the ways are dry they will become roughened and spoiled.

Questions on Lathe Construction—I

1. What are the ways used for? How are they shaped? How are they finished? Why?

2. Explain how the carriage is moved along the ways by hand. What is the feed rack? What is the feed-rack pinion? Why is it called a pinion?

3. How are the ways cleaned and oiled properly? What will occur if they are allowed to become dry?

4. Where is the "live center" located? Where is the "dead center" located?

5. Why are the centers called "live" and "dead"? Which is hard? Which is soft? Why?

6. Move the tailstock along the bed. What other lengthwise adjustments may be given the dead center?

7. If through carelessness the tail spindle is run off the screw, what precaution must be taken regarding the keyway?

8. How is the dead center removed? What caution must be observed?

9. How is the tailstock adjusted sideways? Why is it necessary to first loosen the clamping bolts?

10. How is the tail spindle tightened? Will a quarter of a turn of the locking lever loosen it?

11. Where is the main spindle of the lathe? Why must it be substantial and accurate? Why must the bearings be substantial and accurate?

12. What establishes "the center line of a lathe"? What is it parallel to? When is the dead center "in line"?

13. How is the live center removed?

14. What are two advantages of the hollow spindle?

15. What part of the carriage is called the saddle? The apron? The tool rest?

16. How is the top of the saddle finished? Why must it be kept clean and well oiled?

17. Can you move the carriage by hand when the split nut is closed? When the feed-control knob is tightened? When the carriage clamping screw is tightened? Give reasons.

18. Why does a machinist, before starting to work on a lathe, always try the carriage to make sure it runs freely?

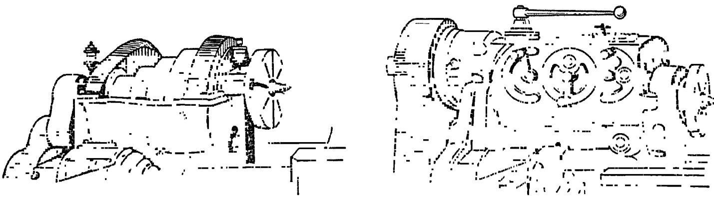
19. How is motion transmitted from the main spindle to the feed rod? To the lead screw?

20. Describe the action of the split nut. Why is it called a split nut?

21. How are the gears in the apron oiled?

SPINDLE SPEEDS

24. How Different Speeds Are Obtained.—Because of the wide difference in diameters that may be turned in a lathe, it is necessary to have the spindle revolve at different speeds in order to obtain the proper cutting speed[1] for any size of



(a) (b)

Fig. 15.—Types of headstocks: (*a*) cone-pulley and back-gear drive; (*b*) geared head.

work, from a very small diameter to the largest that the lathe will swing.

One of the oldest methods of obtaining speed changes is by having the power delivered by a belt from overhead to a pulley with different diameters or "steps" called a *cone pulley*. The cone pulley may be used alone, as in bench machines for example, or in connection with back gears as in the general machine tools (see page 45).

In the high-duty machines this cone-pulley drive has given way to the more modern single-pulley "geared head." In this drive the power is delivered from overhead to a wide single-faced pulley or from its own individual motor through a silent chain or multiple V belt, thence through various "gear runs" to give the different spindle speeds. The power is controlled by a powerful friction clutch and brake between

[1] Cutting speed, see page 96.

the pulley and the driving shaft. The drive is not unlike an automobile drive. The automobile has one high speed (no gears), with one reverse and two additional forward speeds through gears, all controlled through a clutch and "gear-shift" lever. A modern lathe, for example, may have 20 "selective speeds" all through changes of positions of "sliding gears" within the headstock, that are controlled by two or three convenient "speed-change" levers.

There follow illustrations and descriptions of pulley and gear drives, then examples of the application of these principles in the lathe. And the same principles are used in all other kinds of machine tools. There is more than one lesson, probably more than a dozen lessons, in this chapter. It contains some of the very necessary theory of machine construction. Be sure to study this section with the idea of *reasoning* and *understanding;* make sketches if it is easier to make sure. If this theory is learned lesson by lesson, step by step, it will be found interesting all the way, and helpful always.

Since means have been found of making gears strong, long lived, with no chatter, and especially quiet, they have become most important in mechanical drives. The quiet-running geared head with many spindle speeds, easy control, and full power is a distinct development. There are probably a thousand gears made today to one made 25 years ago. But this doesn't mean that line shafts and countershafts, pulleys and belts are obsolete; they have their place too, and a big place, in machine shops and factories.

25. Countershaft.—In a shop with belt-driven machinery, the power is transmitted by belting from the engine or motor to a "line shaft," possibly as long as the shop and with dozens of pulleys. From a line shaft pulley the power may be transmitted to the "countershaft," thence to the machine.

In some drives the countershaft is omitted; the power comes from the line shaft pulley direct to a wide-faced pulley on the machine. Motion from this pulley to the "geared head" of the machine is controlled by a friction clutch in the head unit.

Figure 16 shows one design (Reed) of countershaft with two friction pulleys (or loose pulleys) and a cone pulley. The countershaft runs in bearings which are supported by hangers which are fastened to the stringers above. It is essential that the countershaft is level and also that it is parallel to the line shaft, or the belts from the line shaft to the countershaft will not stay on when the machine is working.

The cone pulley is tightened to the shaft by means of a set-screw. Each friction pulley is normally free on the shaft

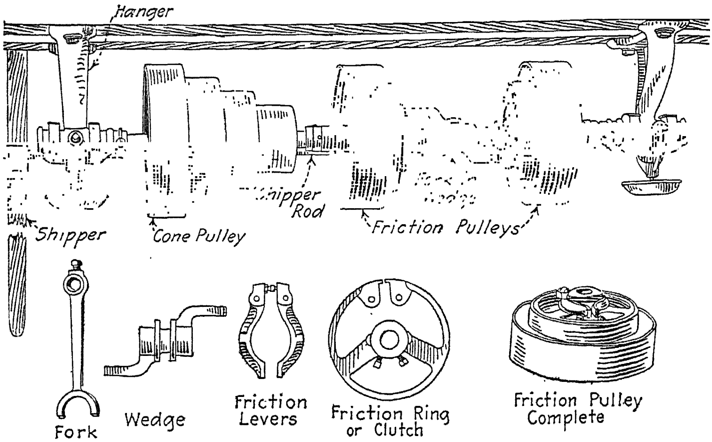


FIG. 16.—Countershaft.

while its friction ring is tightened to the shaft by two set-screws. The friction ring is finished on the outside a trifle smaller than the diameter of the finished inside of the pulley rim. It will be observed that the ring is split between the two lugs. The friction levers are fastened to these lugs and adjusted so as to expand the ring sufficiently to hug the pulley when the wedge is forced between them. The position of the wedge is controlled by the fork fastened to a rod to which is attached the "shipper." One of the belts from the line shaft is a "crossed" belt while the other is an "open" belt, consequently the loose pulleys run in opposite directions. The position of the shipper, therefore, determines whether the

countershaft (and the machine) shall be (1) idle, (2) running forward, or (3) reverse. The reverse is seldom used except when cutting threads.

26. Changing Belts.—Learn as soon as possible to change belts. It is a knack that is part of the machinist's trade.

CAUTION: *Always be sure, before changing a belt by hand, that there are no hand-ripping metal fasteners in the belt.*

When changing the belt from one cone to another remember that the belt "leads" from one pulley to another. As illustrated in Fig. 17 the belt at C is leading from pulley A to pulley B, and at D it is leading from B to A.

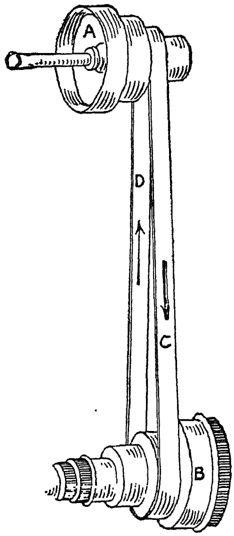


FIG. 17.

A belt is controlled in its leading direction; thus a pressure at C will lead the belt off the pulley B, and a pressure at D will lead the belt off the pulley A.

It is good practice always to throw a belt off with a wrench handle or a stick, but it is often necessary to throw it on by hand. Get fairly close to the pulley, use the flat of the hand, hold the fingers stiff and close and never curl them around the edge of the belt or they might get pinched.

Suppose it is required to "speed up" a lathe, the operator will run the belt down to the smallest step on the lathe pulley, and by putting his right hand inside the belt and pulling it steadily towards him will speed up the cone pulley; then with a slight pressure in the proper direction and with a tossing movement of his left hand, he will find it easy to place the belt on the larger step of the cone pulley on the countershaft.

Sometimes if the belt is fairly tight it is well to use a belt pole,[1] but with a little experience one can usually place the belt

[1] *Belt Pole.*—A pole of sufficient length to reach the line shafts and countershafts, with a pin or stud 4 or 5 in. long fastened at one end at right angles to the pole. Used to push belts on or off.

on any step in a few seconds. It is often advisable to have the back gears and the lockpin both *out* when changing the belt because this allows the driven pulley to run free and makes the change much easier. (For lockpin see page 44.)

NOTE.—For description of how belts are fastened see Appendix (page 373).

SPEEDS OF PULLEYS AND GEARS

The beginner should consider carefully the construction of the driving mechanism of the lathe, and understand thoroughly the method of obtaining the different speeds, because similar mechanical principles are involved in nearly every sort of machine tool. The first thing to understand when beginning

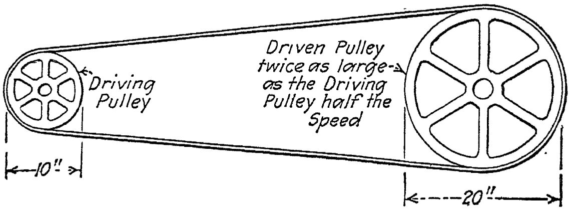


FIG. 18.—Driving and driven pulleys.

a study of machine speeds is the driving and driven action of pulleys and gears.

27. Driving and Driven Pulleys.—When two pulleys are connected by a belt, motion is transmitted from the driving pulley to the driven pulley.

If the driven pulley and the driving pulley are the same diameter the driven pulley will make as many revolutions as the driving pulley, because the distance that the belt is carried along by frictional contact with the driving pulley during one revolution is equal to the circumference of the driving pulley, which is equal to the circumference of the driven pulley. If the *driven* pulley is *twice* the diameter of the driving pulley (Fig. 18), the driven pulley goes half as fast as the driving pulley, because the distance that a point on the belt is carried along by frictional contact with the driving pulley during one revolution is equal to *half* the circumference of the driven

pulley. If the *driven* pulley is *one-third as large* in diameter as the driving pulley, it will revolve three times as fast as the driving pulley because the circumference of the driving pulley is three times the circumference of the driven pulley. That is, the speeds of driving and driven pulleys are to each other *inversely* as their diameter.[1]

28. Driving and Driven (Follower) Gears.—The same reasoning is true with gears with different numbers of teeth as with pulleys of different diameters. If a driving gear having 30 teeth (Fig. 19) is in mesh with a gear having 60 teeth, when the driving gear has made one full revolution it has engaged only 30 of the 60 teeth of the follower gear and turned it only half around. If a driving gear *D* has 40 teeth (*a*, Fig. 20), and the follower gear *F* has 20 teeth one revolution of the driving gear will revolve the follower gear two revolutions. That is, the velocities of the driving and follower gears are to each other *inversely* as the numbers of their teeth.[2]

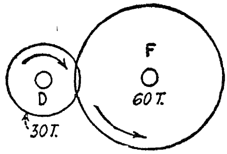


FIG. 19.—Driver and follower gears.

29. Simple Gear Train.—Two gears in mesh are called a pair of gears. Three or more gears, the first meshing with the second, the second with the third, and so on, constitute a simple train of gears. The gear between the driving and follower gears in a simple train is known as an "idler" or "intermediate." A small gear in a pair or a train or gears is often called a "pinion."

Placing an intermediate gear (of any number of teeth) between two gears changes the direction of rotation of the follower gear but does not change the velocity ratio. This is illustrated in *b*, Fig. 20. One revolution of *D* will engage 40 teeth in *I* (no matter how many teeth *I* has), and *I* will engage 40 teeth in *F* turning it around twice, the same as in *a* without the intermediate. Note, however, that in *b* the direction of rotation of the follower gear *F* is changed.

[1] For formulas on speeds of pulleys see Appendix (page 369).
[2] For formulas on velocities of gears see Appendix (page 371).

Any number of intermediates will not affect the *velocity ratio* between the driving and the follower gears but the *direction of rotation of the follower gear depends on the number of*

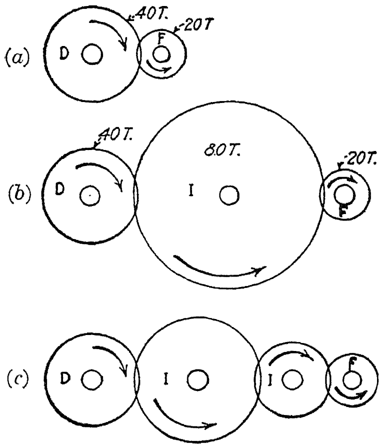


FIG. 20.—Simple gear trains. (*a*) No intermediate; (*b*) one intermediate; (*c*) two intermediates.

intermediates; thus with one intermediate the follower gear will revolve in the same direction as the driving gear (*b*, Fig. 20) and with two intermediates the direction will be reversed (*c*, Fig. 20).

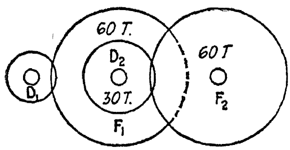


FIG. 21.—Compound gear train. Gears F_1 and D_2 are mounted on the same shaft and serve as the compound between D_1 and F_2.

30. Compound Gear Train.—If, however, there are two gears fastened to the same shaft, or to a quill, or fastened together in any way so that when one revolves the other must revolve at the same speed, one engaged by the driving gear and the other engaging the follower, these two gears *mounted on the same shaft* are not intermediates. They are respectively the follower gear of the first pair and the driving gear of the second pair of four gears which form a *compound gear train.* In a compound gear train the sizes of the gears between the first driver and the final

follower cannot be disregarded as in a simple train. This is illustrated in Fig. 21. Suppose D_1 revolves twelve times, F_1 will revolve four times being three times as large; D_2 will revolve four times because it is fastened to the same shaft as F_1, and the final driven gear F_2 will make *two* revolutions.

Questions on Pulley Speeds and Gear Velocities

1. What is meant by the velocity of a gear?

2. The velocities of two gears in mesh (with teeth engaging) varies inversely as the numbers of teeth. What do you mean by "inversely"?

3. A driving gear *D* has 60 teeth and meshes with a gear *F* of 40 teeth. How many times will *F* (the follower gear) revolve when *D* makes 10 revolutions? Why?

4. Introduce an intermediate gear *I* of 120 teeth between *D* and *F*. How will this affect the result as to the relative speed of *F*? As to direction of *F*?

5. Introduce one more intermediate of any number of teeth in this train of gears. What will be the result as to relative speed of *F*? As to direction of *F*?

6. What is meant by a simple train of gears?

7. What does the introduction of one or more intermediates serve to do as regards the speed of the follower gear? As regards direction of follower gear? Is this always true in any simple train of gears?

8. The reverse of the lathe is often faster than the forward motion. What causes this?

9. When installing a lathe it is desired to have the slowest speed of the cone pulley 100 r.p.m. The largest step of the cone pulley on the lathe is 10 in. in diameter, the smallest step on countershaft cone pulley is 6 in. in diameter, and the loose pulley is 12 in. in diameter. What diameter pulley will be required on the line shaft which runs at 250 r.p.m.?

10. A pulley 12 in. in diameter is running at 220 r.p.m. and is connected by a belt to a pulley 8 in. in diameter. How fast does the smaller pulley revolve?

11. What is meant by an inverse ratio?

31. Direct Spindle Speeds.—There are usually two or more series of speeds in all except the smallest sizes of machine tools. They are commonly known as the *direct* speeds or "*back gears out*" and *indirect* speeds or "*back gears in.*"

Different *direct* speeds in a belt-driven (cone pulley) lathe are obtained by changing the driving belt to the various steps

on the cone pulley. The cone pulley of the lathe is the *driven* pulley and changing the belt from a larger to a smaller step on this pulley (and a corresponding larger step on the counter-shaft cone pulley, which is the driving pulley) increases the speed.

The cone pulley (Fig. 23) is not fastened to the spindle but may revolve freely upon it. The spindle-driving gear D called the "face gear" is keyed to the spindle, and to cause the spindle to revolve, it is necessary to transmit the motion from the belt-driven cone pulley to the face gear D. The *direct* speeds are obtained by locking together the cone pulley and the face gear D by the *lockpin* L. As many different direct speeds may be thus obtained as there are steps on the cone pulley.

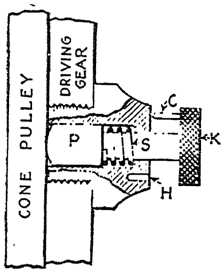


FIG. 22.—Lockpin. When the lockpin is arranged as shown the cone pulley is free to revolve past the face gear (spindle-driving gear). There are three or more equidistant holes in the pulley which makes it unnecessary to turn the pulley more than one-third of a revolution to bring a hole in front of the plunger.

32. Lockpin (Fig. 22).—The plunger P is really the locking pin; when it enters the cone pulley it locks the face gear to the pulley.

The spring S tends to push the plunger into the hole in the cone pulley but in the position shown in Fig. 22 is kept from doing so by the pin C. To "put in" the lockpin, turn the knurled knob K until the pin enters the hole H and pull the belt *by hand* until one of the holes in the pulley comes in front of P and P enters the hole. *Never* start the machine by power until the plunger is in the hole of the pulley. When "taking out" the lockpin pull the knob K until the pin is out of the hole and turn the knob part away around.

33. Back Gears.—In order to get a large number of different speeds, many engine lathes and other machine tools are equipped with back gears. The function of the back gears is to give a speed to the spindle which is slower than the speed of the cone pulley. This reduction in speed also gives a

corresponding *increase* in power. Some lathes have two or more sets of back gears.

Referring to Fig. 23 the back gears *B* and *C* are both fastened to the quill[1] which revolves on the shaft *E* which is supported by brackets back of the spindle. It will be noted that the ends of this shaft, the bearings, are eccentric (out of center) with the part of the shaft on which the quill revolves. This construction is for the purpose of changing the position of the back gears, putting them in to engage with the gears *A*

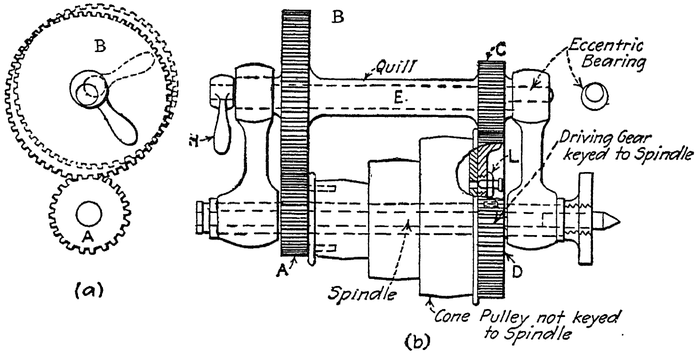


FIG. 23.—Lathe headstock. In the cut (*b*) shows a view looking down on the headstock with the back gears supported back of the cone pulley; (*a*) is an end view and shows the two positions of the back gears and back-gear handle, heavy line "in" light line "out."

and *D* or taking them out of mesh by partly rotating the shaft by means of the back-gear handle *H*.

34. Indirect Spindle Speeds, Back Gears (an application of compound gearing).—It will be observed (Fig. 23) that the back gears are in and the *lockpin* is out. Gear *A* is fastened securely to the cone pulley and power is transmitted from the cone and gear *A* to back gears *B* and *C*, and from *C* to face gear *D* which is keyed to the spindle. If gear *B* is three times as large as *A* it will revolve one-third as fast. *B* and *C* are both fastened to the same quill and revolve at equal speeds. If *D* is three times as large as *C* it will revolve one-third as fast,

[1] *Quill.*—Hollow sleeve which revolves on a shaft and carries pulleys, gears, clutches, etc.

with the result that D will revolve one-third of one-third or one-ninth as fast as A. There are, of course, as many back-gear speeds as there are steps on the cone pulley. A lathe with three steps on the cone pulley and with back gears would thus have six spindle speeds, three direct and three indirect.

35. Revolutions per Minute (R.P.M.).—In order that the beginner may realize the different revolutions per minute (r.p.m.) of the lathe spindle at each of the positions of the

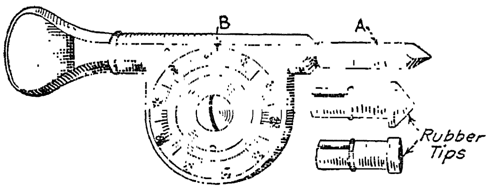


FIG. 24.—Speed indicator. One hundred turns of the spindle A cause one revolution of dial B which is graduated in one hundred divisions and every ten divisions numbered. Speeds too fast to count mentally are easily obtained. Rubber tips may be applied to the indicator spindle.

speed-change levers (or of the belt on the different steps of the cone pulley if belt driven), he may ascertain the different speeds by counting the lower number of revolutions, and by using a speed indicator (Fig. 24) for the faster speeds (get the number of revolutions for ½ min. and multiply by 2 to get the r.p.m.).

36. Sliding Gears.—To obtain a number of speeds of a driven shaft, or of a spindle of a modern machine tool, a construction known as *sliding gears* is much used. Suitable levers are arranged to operate a fork as in *a*, Fig. 25, or a yoke as in *c*, to slide the cluster[1] of gears to the position desired.

[1] Definitions:

Gear Cluster.—A number of gears mounted together to slide as a unit.

Key.—A piece, usually rectangular section, used to fasten to a rod or shaft a pulley or gear or similar part having a hole fitting the shaft, to keep this part in place and to keep it from turning.

Keyway (or key seat).—The groove in the rod or shaft, or in the hole in the pulley or gear, to fit the key.

Spline.—A comparatively long keyway in the shaft or rod to permit sliding of the gear if desired.

Until quite recently one sliding key, or "feather," was all that was needed to drive almost any gear, or bank or cluster of gears, but since so much more power must be delivered by

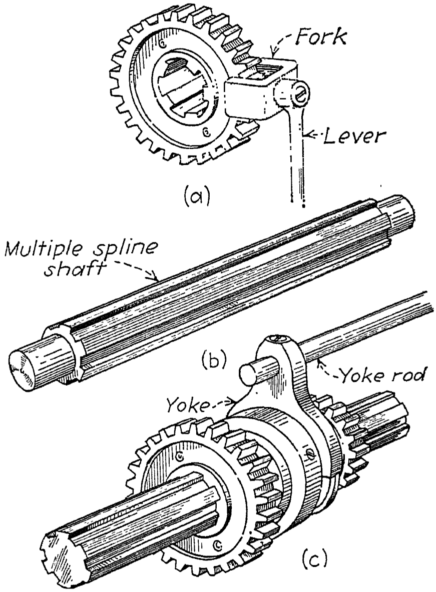


FIG. 25.—(*a*) Multiple spline gear and fork for moving gear. (*b*) Multiple spline shaft sometimes called integral key shaft. (*c*) Yoke for moving gear.

the heavier, faster machines, the heat-treated alloy-steel gears that are now used must have more than one key, and the shaft with the *multiple spline*, sometimes called *integral key*, is commonly used (see *b*, Fig. 25).

An idea of the importance of this design may be gathered from the fact that standards for broached-hole dimensions from $\frac{3}{4}$ to 3 in. diameters have been established; for permanent fit, to slide when not under load, and to slide when under load.

Feather.—A sliding key, fitting tight in the sliding part and easy in the spline.

Integral.—Made in one piece.

Multiple.—More than one, usually several.

Broaches for making the holes and cutters for hobbing the shafts are manufactured.

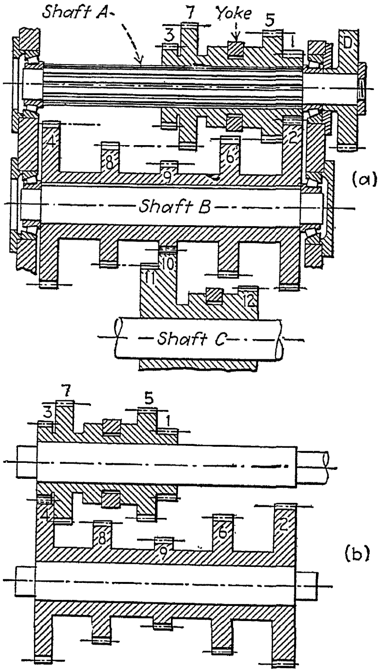


FIG. 26.—Sliding gears. The diagram is to show the method of obtaining four speeds of a driven shaft *B* by sliding a cluster of gears 1-5-7-3 to four positions. Gear *D* drives the multiple-splined shaft on which is the gear cluster. In the position shown in *a*, gear 1 engages gear 2 and gives the slowest speed because 1 is the smallest driving gear and 2 is the largest driven gear. In *b* is shown the position of the sliding gears for the next faster speed, with gear 3 engaging gear 4. Similarly if the cluster is moved to engage gear 5 with gear 6 the speed will be still faster, and when the gears are moved to engage 7 and 8, the fastest of the four speeds is obtained.

To go a step farther, the gears 10, 11, and 12 (shown in (*a*) only) are sliding gears on shaft *C*. With gears 9 and 10 engaged as shown there are four speeds of shaft *C* obtainable as explained above. Engaging 11 with 8 will give four more speeds of *C*, and with 12 and 2 engaged there are four more making a total of twelve speeds for shaft *C*.

The diagrams in Fig. 26 show a portion of a sliding-gear arrangement and illustrate, first, the manner of using one cluster of sliding gears to obtain *four* changes of speed. Possi-

bly it will be interesting to look further at the diagram and note the arrangement of three more sliding gears just as simple, to get a total of *twelve* speeds.

THE OPERATION OF LATHE FEEDS

37. The Feeds of the Lathe.—The regular or longitudinal feed of the lathe is the travel of the whole carriage along the ways of the lathe parallel to the axis of rotation of the main spindle, that is, parallel to the center line of the lathe. The cross feed of the lathe is the travel of the cross slide at right angles to the center line of the lathe.

Each of these feeds may be operated in either direction by hand (called hand feed) or automatically in either direction (called power feed). The amount of the hand feed is controlled absolutely by the hand of the operator: it may be slow or fast, and the lathe may or may not be running. The power feed, however, is an automatic movement of the tool carrier of a desired definite amount for each revolution of the spindle. Motion is transmitted from the revolving lathe spindle through gearing to the feed rod, and from the revolving feed rod through the apron gears to the carriage, or to the cross-feed screw, as desired. This mechanism is interesting because it involves the use of gears, clutches, and certain other mechanical principles.

38. The Tumbler-gear Train and the Change-gear Train.—Fig. 27 illustrates the gearing from the spindle to the feed rod. The gear *Sp* is keyed to the spindle and transmits motion through the tumbler gear R_1 to the inside stud gear or fixed stud gear *F.S.* which is keyed to the stud shaft. (The use of the gear R_2 will be explained presently.) This train of gears is called the tumbler-gear train or sometimes the reverse-gear train and is usually within the headstock casting.

The stud gear *St* is a change gear, that is, if desired it may be removed and a larger or smaller gear put on in its place. Since it also is keyed to the stud shaft it revolves when the fixed stud gear *F.S.* revolves. The stud gear transmits

motion through the intermediate gear I to the screw gear[1] Sc which is also a change gear. Sc is keyed to the lower feed-box shaft and motion is transmitted from this shaft to the upper feed-box shaft by the gears E and B. When the clutch C is in the position shown in the diagram motion is transmitted

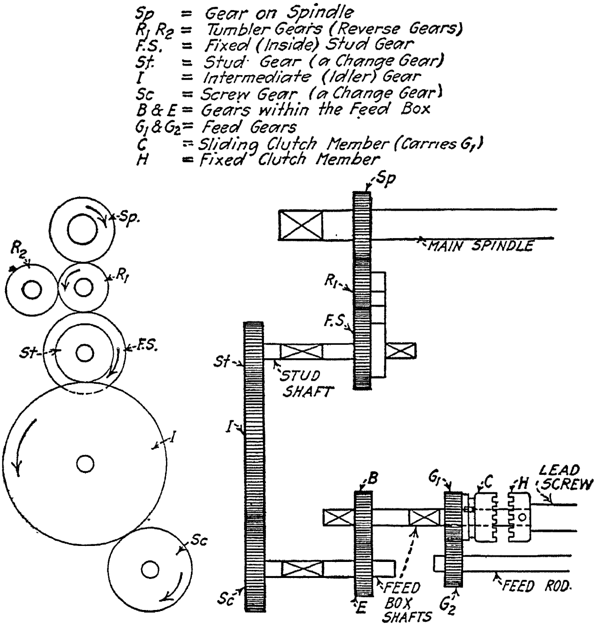


FIG. 27.—Gearing from main spindle of lathe to feed rod and lead screw.

from the upper feed-box shaft to the feed rod by the feed gears G_1 and G_2.

When the sliding-clutch member C is moved to engage with the fixed-clutch member H, the lead screw is caused to revolve and the feed rod stops because the feed gears G_1 and G_2 are no longer engaged.

[1] *Screw Gear.*—In the older lathes the screw gear was keyed direct to the lead screw, hence its name. It was used only for thread cutting. In such lathes *feed* motion was transmitted by a belt from a cone pulley on the stud shaft to a cone pulley on the feed rod.

39. The Tumbler Gears.—The operation of the tumbler gears R_1 and R_2 is illustrated in Fig. 28. These gears are carried on a bracket which is pivoted on the stud shaft and operated by means of the reverse-gear handle (44, Fig. 13), to any one of the three positions shown. It will be observed that the two tumbler gears are intermediate gears between the spindle gear and the fixed stud gear, and are so mounted on the bracket as to make it possible for the operator to have either one intermediate or two intermediates in mesh between the driven gear and the driving gear, or to throw them both

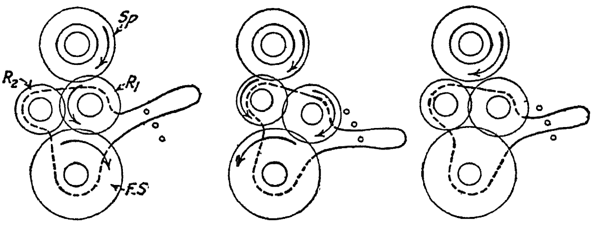


FIG. 28.—Illustrates operation of reverse gears or tumbler gears.

out of mesh with the driving gear. That is with the three positions of the reverse-gear handle the stud shaft may have (1) forward movement, (2) reverse, (3) no motion. The function of the tumbler gears is to reverse the direction of revolution of the feed rod when turning or of the lead screw when cutting threads. That is, in either case the tumbler gears serve to *reverse the direction of the carriage*. Do not forget this feature.

40. The Intermediate Gear in the Change-gear Train.—The bracket or "quadrant" which carries the intermediate gear is arranged to pivot on the stud shaft through a certain distance (see Fig. 29). The stud on which the intermediate gear revolves is adjustable to any position in a fairly long slot in the bracket. With these two adjustments it is possible to arrange the intermediate gear to engage both the stud gear and the screw gear no matter what size these gears may be.

41. Quick-change Gears.—All modern lathes are equipped with what are usually termed "quick-change gears." In such

lathes it is not necessary to change the gears on stud and screw (except in rare cases) to give the different feeds desired, or to cut the different pitches of threads. The change is made by merely shifting the position of one or more handles.

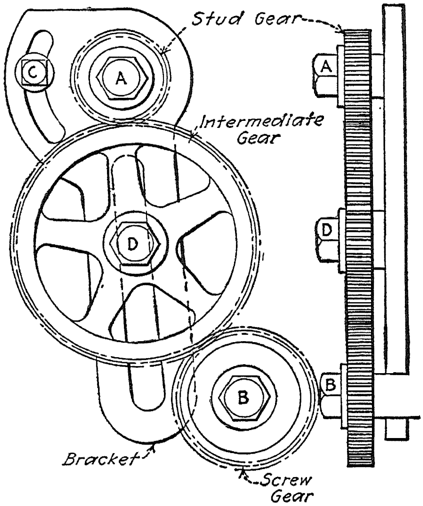


Fig. 29.—Change-gear train showing intermediate gear bracket or "quadrant." When changing gears, first put reverse handle in neutral to avoid any chance of the gears moving if the lathe should be started accidentally. Then loosen the nuts *A* and *B*, then the binding screw *C*, and finally the nut *D*, being careful not to let the intermediate fall too hard against the bottom of the bracket. When the gears on the stud or screw or on both are changed as desired, lift the intermediate to engage the stud gear, pinching a piece of paper between them to allow for clearance, and tighten *D*, then swing the bracket to pinch paper between the intermediate and the screw gear and tighten *C*.

Always put on the gears with the numbered side out so that if necessary the number of the teeth may be determined quickly. When removing the gears have the key on top so it will not fall out. The change gears should fit easily; the nuts should go on easily and should finally be tightened with a wrench.

The diagram (*d*, Fig. 30) illustrates the gearing by means of which 36 different feeds and also 36 different threads may be obtained by moving two handles. One handle serves to move the feathered pinion *K* along the shaft *S*3 and also to engage a rocker gear *L* with any gear in the cone of gears (in the diagram the rocker gear is shown engaged with the 36-tooth gear). All

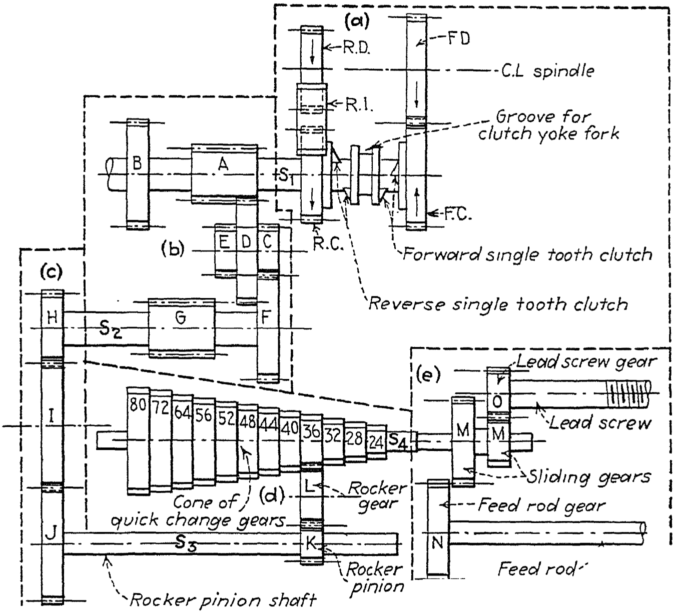


FIG. 30.—Shows the gearing from the main spindle to feed rod and lead screw. The dotted lines dividing the sections *a*, *b*, *c*, *d*, and *e* are for clearness.

(*a*) Gears and clutch for forward and reverse motion of reverse shaft *S*1. *F.D.* forward drive gear; *F.C.* forward clutch gear; *R.D.* reverse drive gear; *R.I.* reverse intermediate gear; *R.C.* reverse clutch gear. (Remember that adding a gear in a train serves to change the direction of the driven gear.)

(*b*) Ratio gears for obtaining three speeds of the stud shaft *S*2 by sliding the gear cluster *EDC*. The arrangement for the slowest speed is as shown, *A-D-C-F*. The middle speed arrangement would be *A-D-G*, for fastest speed the gear run would be *B-E-D-G*.

(*c*) The change gears, called *stud gear H* and *screw gear J* and *intermediate gear I*. (For additional speeds of *S*3 reverse the positions of *H* and *J*. For 11½ threads per in. a 69-tooth gear is provided.)

(*d*) Cone of gears any one of which may be connected to the rocker pinion *K* on the shaft *S*3 by moving the rocker gear *L* and pinion *K* to the position indicated on index plate for the given thread or feed, and engaging in the given place.

(*e*) Gears for connecting feed rod or lead screw to the cone gear shaft *S*4. The sliding gears *M* are moved to the left to engage the feed rod gear *N*, to the right to engage the lead screw gear *O*.

of the 12 gears of the cone of gears are keyed to the feed-cone shaft $S4$, therefore with a given speed of $S3$ any one of 12 different speeds of $S4$ may be had, because each one of the cone of gears is, when engaged, a follower gear of a different size. Three speeds of $S3$ are obtained by moving a handle to slide the cluster $E\ D\ C$ to each of its three positions. Therefore 3 speeds of $S3$ multiplied by 12 speeds of $S4$ gives 36 speeds of the feed rod or of the lead screw. Whether the feed rod or the lead screw revolves depends upon the position of the sliding gear M, engaged with N to revolve the feed rod, or with gear O to revolve the lead screw.

Index plates indicate the position of the levers for "feed" or "thread" (thread is sometimes indicated as "screw" or "lead") and also show the positions of the other levers for each pitch of thread and the corresponding amount of feed.

42. Reversing Gears.—In paragraph 39, page 51, the tumbling-gear reverse was discussed. Another much-used method is shown in *a*, Fig. 30. In the position shown, with the clutch neutral and the driven gears free, the shaft $S1$ does not turn. The clutch, feathered on the shaft, is moved to the right to engage *F.C.* for the forward motion of the driven shaft, to the left to engage *R.C.* for the reverse direction. Note that gears *F.D.* and *F.C.* are the same size, also that gears *R.D.* and *R.C.* are equal, therefore driving and driven speeds are equal, either forward or reverse. The size of the intermediate *R.I.* is unimportant; it serves to revolve the gear *R.C.* in the opposite direction of gear *F.C.*

One special advantage of this reversing mechanism, with a single-tooth clutch as shown, is that the exact relative positions of the driving and driven shafts are constantly maintained. The value of this feature in thread cutting is explained in paragraph 209, page 248.

43. The Apron.—The action of the gears in the apron is shown in Fig. 31, and Figs. 33 and 34 show front and back views of the more rigidly constructed apron. Figure 36 shows the quick acting, lever-operated friction clutch which is replacing the older knob-turning type.

There are many designs of aprons, some with single-bevel-pinion drive (Fig. 31), some with two bevel pinions, one for reversing the direction of the feeds (Fig. 33). Other designs have a worm-and-worm-wheel drive. The point is, that while they differ in design, all serve the same three purposes—to transmit, by means of gears, the motion from a splined feed rod to either the *feed-rack pinion* which meshes with the feed rack to feed the whole carriage (long feed), or to the *cross-feed pinion* which is keyed to the *cross-feed* screw; and also, by closing the split nut on the lead screw, to transmit positive motion to the apron for the purpose of cutting threads.

If the beginner in the shop gets a conception of the *principle* underlying apron construction, it will be easy to understand any design. This is true of any attachment or unit, as, for example, the lathe taper attachment, the milling-machine index head, or the ratchet and pawl for shaper feed. It is not difficult to see through the principle of almost any mechanism, that is, what the purpose is and how it is accomplished, then follows interest in details of design.

44. The Apron Mechanism.—The way in which motion is transmitted from the revolving feed rod through the apron gears, to cause either the longitudinal (long) feed or the cross feed to operate, is shown in Fig. 31. This cut shows an older type of apron; the later stronger designs (see Fig. 33) have "double-wall construction" or "box section" to provide outer bearing supports for all studs (a gear with a bearing on each side is more rigid and has smoother action than if it is supported by only one bearing). However, the principle of the mechanism is the same for both, and Fig. 31 being open is easier to see and understand.

The feed pinion (7) engages the feed rack (35, Fig. 13) and when it revolves causes the carriage to move along the ways (long feed). The gear (10) engages a small pinion which is keyed to the cross-feed screw, and when it revolves causes the cross-feed screw to turn. The other gears in the apron are used to transmit motion either from the feed rod to gear (7)

to give long feed, or to gear (10) to give cross feed, and are thus explained:

Motion is transmitted from the feed rod to the bevel pinion (1). (The feed rod has a long spline—keyway—and the pinion is feathered on the feed rod. The carriage may be moved along the ways of the lathe and the pinion may slide over the feed rod but when the feed rod turns the bevel

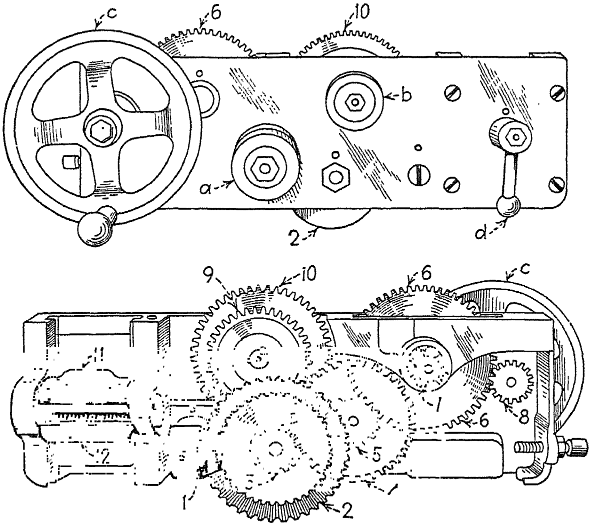


FIG. 31.—Older type of lathe apron.

pinion turns.) The pinion (1) turns the bevel gear (2). The small spur gear (3), shown in dotted lines, is fastened to the bevel gear (2) and meshes with gear (4) and gear (4) meshes with gear (9). Therefore when the feed rod revolves the gears (1), (2), (3), (4), and (9) revolve, whether or not either feed is operating.

To obtain the longitudinal feed tighten the feed knob *a* which operates the friction[1] between gear (4) and gear (5)

[1] *Friction Clutch.*—The cone friction is used. The tapered cone of one member fits the corresponding taper cup in the other member and drives by friction. See parts 4 and 5, Fig. 36, page 60.

(the latter shown in dotted lines) and causes (5) to revolve with (4). Gear (5) engages gear (6) and when (6) turns it causes the feed pinion (7) to move and thus feeds the carriage.

To obtain the cross feed tighten the cross-feed control knob *b*. This tightens the friction between gear (9) and gear (10) and as stated above, gear (10) meshes with a small pinion (not shown) fastened to the cross-feed screw.

The handwheel *c* operates the pinion (8) which meshes with (6). Therefore when the handwheel is turned it causes a movement of (8) to (6) to (7) and moves the carriage.

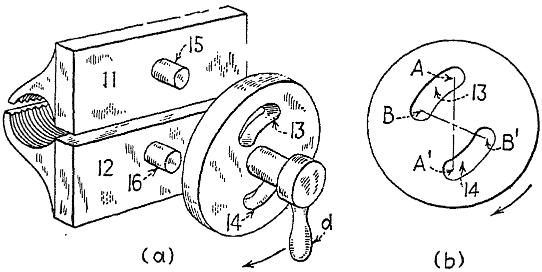


FIG. 32.—The two halves of the split nut and the cam slots in the disk which serve to "open" or "close" the split nut. It will be noted in the diagram (*b*) that when the pins (15) and (16) are in positions *B* and *B'*, they are closer together than when in positions *A* and *A'*.

The two halves of the split nut (11) and (12), Fig. 32 and also Fig. 31, are operated by the handle *d* through the cam slots (13) and (14) in the disk. The pins (15) and (16) which lie in the slots are caused to move toward each other when the handle is moved in the direction of the arrow and away from each other as the handle is moved back.

45. High-duty Apron.—A type of apron that is very popular in lathe construction is illustrated in Figs. 33 and 34. One feature is the bevel-gear feed-reversing mechanism. It will be noted that instead of one bevel pinion feathered on the feed rod this apron has two, one on each side of the large bevel gear. The engagement of the pinions with the bevel gear is controlled by a handle on the front of the apron (3, Fig. 34). Each pinion may serve to revolve the gear but being

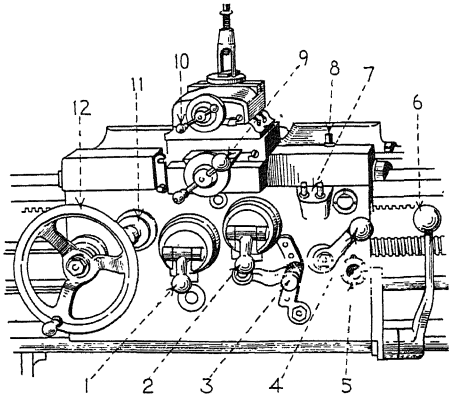


Fig. 33.—Lathe apron. 1, longitudinal feed control lever; 2, cross-feed control lever; 3, bevel-gear reverse lever [now in neutral, in which position the split nut (half-nuts) may be closed]; 4, split nut handle; 5, chasing dial (used when cutting threads); 6, control handle; (carried with the apron) operates the clutch and brake in the main drive to start and stop the machine (in some machines operates the one tooth forward and reverse clutches to start and stop the lead screw in either direction); 7, one-shot oiling reservoir; 8, carriage clamping screw; 9, cross-feed handle; 10, compound-rest feed handle; 11, handle for disengaging feed-rack pinion; 12, long feed hand wheel. (*Courtesy of The American Tool Works Company.*)

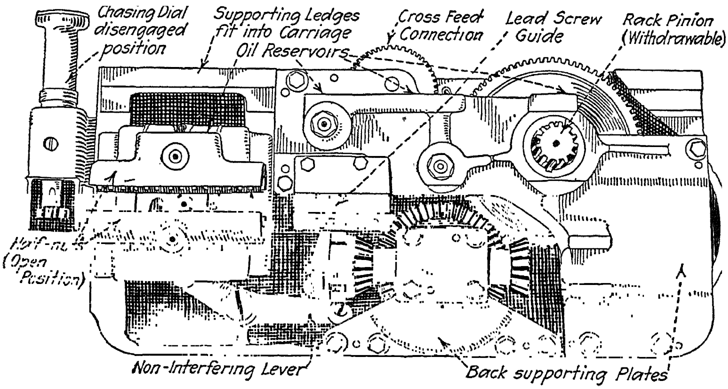


Fig. 34.—Inside view high-duty lathe apron. (*The American Tool Works Company.*) Note the rigid back plate construction; the non-interfering lever; the bevel gears in neutral position. In this view the chasing dial is held vertically, in some models (Fig. 33) it is held horizontally.

on opposite sides revolves it in an opposite direction. The direction of rotation of the bevel gear determines the direction of all the other apron gears and consequently the direction of the feeds. When the handle is in the middle position (as in the illustration) neither pinion engages the bevel gear and there is no feed.

NOTE.—Special attention is called to the bevel-gear reversing mechanism. It is probably the most widely used reversing mechanism in machine construction. It is used in almost every kind of machine tool. Be sure to understand the *principle* of the construction (see Fig. 35).

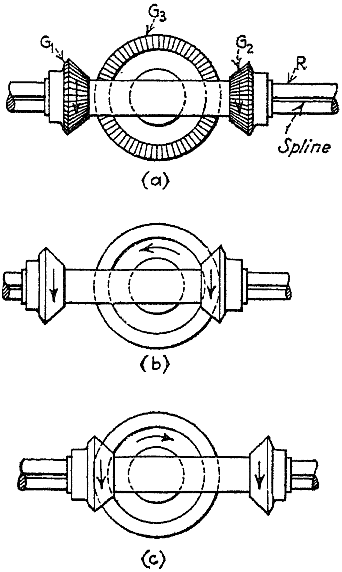


FIG. 35.—Bevel-gear reverse. The bevel pinions G_1 and G_2 are fastened to a sleeve which is feathered on the rod or shaft, R. Gear G_3 is the gear to be driven. In *a* neither bevel pinion is engaged. Moving a lever (3, Fig. 33) will move G_2 to engage G_3 and cause G_3 to revolve in the direction shown in *b*. Moving the lever all the way in the opposite direction will move G_2 out of mesh and engage G_1 with G_3 which will reverse the direction of G_3 as shown in *c*. NOTE: If G_3 were the driving gear and revolved always in the same direction, then the shaft or rod R would be the driven member and its direction would be determined by whether G_1 or G_2 was engaged with G_3.

Another valuable feature is the non-interfering lever. This lever is arranged in such a way as to make it impossible to close the split nut when either bevel pinion is in mesh with the gear. This "fool-proof" construction therefore makes it impossible to throw in the feed when the split nut is closed.

Still another feature on the apron illustrated is the chasing dial which is used when cutting threads without reversing the lathe. This is explained in the chapter on Threads and Thread Cutting (see page 252).

Lathe manufacturers are substituting the drop-lever clutch control for the knob-turning type in the high-duty aprons because the action is quicker.

The clutch control for the longitudinal feed shown in Fig. 36 is used in the lathe illustrated in Fig. 13. The clutch members are held in engagement by a heavy coil spring and disengaged positively and instantly by means of a cam

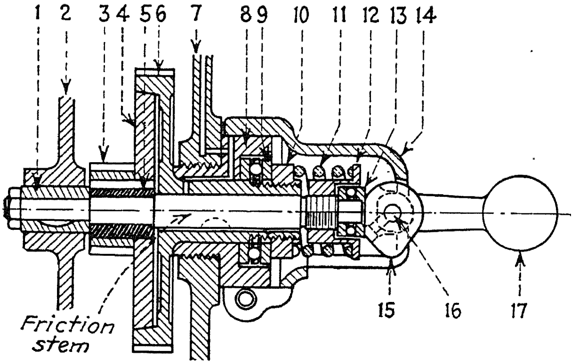


FIG. 36.—Drop lever long feed control. (*The American Tool Works Company.*)
1. Bushing, revolves in back plate. 2. Back plate of apron. 3. Friction pinion. 4. Friction disk. 5. Bushing to which (3) and (4) are keyed. 6. Friction gear. 7. Front plate of apron. 8. Apron bushing. 9. Ball thrust bearing. 10. Retainer nut for ball bearing. 11. Coiled spring for holding tapered clutch surfaces of (4) and (6) in engagement. 12. Adjusting nut for spring tension. 13. Ball thrust bearing. 14. Housing, removable if necessary to adjust the nut (12). 15. Friction lever cam. 16. Friction lever hinge pin. 17. Friction lever ("drop type").

When the friction lever is pushed down, the cam acts against the friction stem thrust bearing (13) and pushes the friction disk (4) away from (6) and "disengages the clutch." When the clutch is disengaged the friction stem revolves in the bushing (5), and the pinion (3) does not revolve. Since the *longitudinal feed gear*, which is keyed to the *feed-rack pinion* is engaged with (3), the feed of the carriage stops when (3) no longer revolves.

actuated by the drop-type control lever. The spring permits of slipping under overloads and thus provides against breaking of gears.

Questions on Lathe Construction—II

1. What is the object of having several spindle speeds (revolutions per minute) in a lathe?

2. What do you understand by cone-pulley drive?

3. Explain the use of a countershaft. Why are there two "loose pulleys"?

4. Explain the action of the friction clutch in the loose pulley.

5. When are the "back gears" used? Explain the eccentric action of the back-gear shaft.

6. Is the lockpin used when the back gears are "in"? When they are "out"? Give reasons.

7. Explain in detail the method of putting the lockpin in.

8. Explain in detail the principle of back-gear action.

9. Make a sketch of the tumbler gears, illustrating how the stud shaft may be driven either forward or reverse.

10. What is the purpose of the intermediate gear in the change-gear train of the lathe?

11. Make a sketch which will show the principle of the action of the cone of gears in the quick-change-gear mechanism of a lathe.

12. Make a sketch which will show the principle of the action of the friction clutch, as used in the apron mechanism.

13. Make a sketch which will show how motion may be transmitted from a beveled gear on the feed rod through gearing to cause the cross-feed screw to revolve.

14. Make a sketch which will illustrate the principle of the bevel-gear reversing mechanism.

46. Other Features of High-duty Lathes.—The use of high-speed cutting tools and the resulting increased power requirements have made necessary in machine design a much stronger construction of the various parts of the machine and a more efficient drive. Keen competition on the part of the manufacturers has developed many interesting types of direct-gear-driven and clutch-controlled mechanisms which give a greater number of spindle speeds than is possible in the cone-pulley type, and also a considerable increase in the power delivered. These machines are differentiated from the cone-pulley type by the terms "geared head," or sometimes "selective gear drive" and "constant speed drive."

Among the advantages of the geared head are the ease and flexibility of the speed changes (by means of levers instead of changing belts) and its adaptability to individual-motor drive, or for driving direct to the single pulley from the line shaft.

With the increase in driving power of the geared-head lathe a corresponding increase in strength and rigidity in the machine as a whole is necessary, particularly in the apron and other

parts of the feeding mechanism. For this reason the high-duty lathe apron is provided with a back plate which affords a back support for the studs, the pinions and gears are made of specially selected steel, and the studs are heat treated and ground.

As has been stated, modern lathes up to 24 in. are provided with the "quick-change-gear" mechanism for feeds and

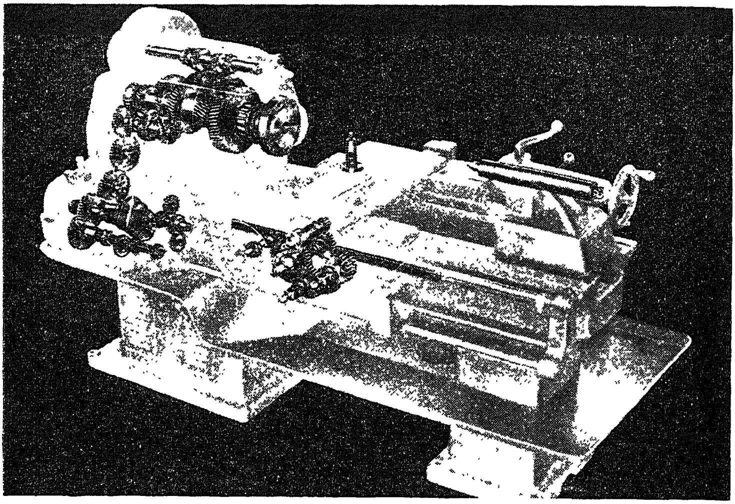


FIG. 37.—Phantom view of a modern lathe. The parts shown in black are of special alloy steels, heat treated. These parts include the main spindle; the tailstock spindle; all gears, studs, and shafts in the headstock, quick-change gearbox and apron; the feed rack and the feed-rack pinion. (*Courtesy of the Monarch Machine Tool Company.*)

thread leads. This mechanism usually provides 36 or more (depending on the make) feeds and as many thread changes, any one of which is instantly available by operating one or possibly two levers. These levers, together with an index plate for showing the position of the levers for the given amount of feed or the pitch of the thread, are conveniently arranged on the gearbox.

A recent development in lathe design is the *flanged* spindle nose to replace the threaded nose for continuous chucking

work or for mounting special fixtures in manufacturing. A centralizing taper is provided on the flange to accurately and quickly locate the chuck plate or fixture, which is then firmly fastened to the flange by bolts and driven by a substantial key. However, for the general run of shop jobs and toolroom work, requiring more frequent changes of faceplates, chucks, etc., the threaded nose is preferred. (It will be observed that the lathe illustrated in Fig. 37 has a flanged spindle nose.)

The original geared heads were noisy and many of them were far from smooth running. These faults have been corrected in many ways:

1. The gears themselves have been greatly improved in shape and finish and durability, hence in quiet and smooth running.

2. The use of too many friction and positive jaw clutches within the head has been avoided by the use of "sliding gears." A simple movement of a conveniently arranged lever serves to engage the mating gears that give the "gear run" for the speed desired.

3. Increased strength, longer life, and much greater smoothness in operation result from having the gears on multiple-spline (integral-key) shafts.

4. Antifriction bearings, roller and ball types, have taken the place of most of the plain bearings.

5. Most machines may be instantly started or stopped by means of a friction clutch and a brake operated by a close-at-hand lever. In some of the modern lathes a forward and reverse control of the *apron gearing* is provided in addition to the friction clutch and brake for the machine spindle.

6. The entire head unit, including the clutch and brake, all bearings, the spindle, the shafts or the sleeves for the gears, are automatically oiled.

The aim has been to improve the appearance of machines, make the speed and feed changes easier, lessen the need of repairs, and, by building the machines in units, make easier any necessary repairs or replacements; also to provide increased

speed and greater strength in order to use up-to-date high-speed cutting tools. And all the while the purpose has been to give the machine *a smoother action* because the good work of any machine depends upon this. To the great credit of the machine-tool designers and builders the increased flexibility and speed of modern machines are combined with a smooth action that gives a high degree of accuracy and an excellent finish in the work produced.

It must not be lost sight of, however, that the lathe-operation essentials are practically unchanged. Certainly the lathe

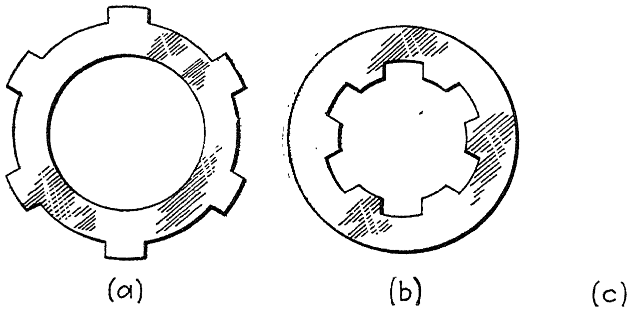


FIG. 38*A*.—Friction clutch disks: *a* outer disk; *b* inner disk; *c* the disks arranged in multiple. Pressing the multiple disks together by suitable cam action serves to produce a frictional contact strong enough to transmit the power. Without pressure the driving disks merely rotate between the others. Which disks are driving or driven depends upon the design of the clutch. Notice the *integral keys* on *a* and the *multiple splines* in *b*.

is faster, and many improvements make for the convenience of the operator, but setting the tool, turning a cylinder, turning a taper, cutting a thread, or boring a hole, and the knowledge, the knack, the skill, the judgment, remain about the same as always.

47. Unit Construction.—By unit construction is meant the building of certain assemblies of parts (units) complete. These units fit exactly into their respective places in the general assembly.

Unit construction in machine-tool building is increasingly used. A machine designed and built under this plan has, in its

manufacture, several advantages in construction and assembly; and, in use, has value as regards adjustments, replacements of worn parts or other repairs, and particularly in its adaptability. Adaptability, in this case, is permitting in the present machine (1) the use of special units or attachments if and when required, and (2) the application and use of improved units as developed, without the expense of a whole new machine.

The unit idea is not altogether new, as witness the apron of a lathe, or the quick-change-gear box, but its further use in the lathe and in other machines is developing rapidly.

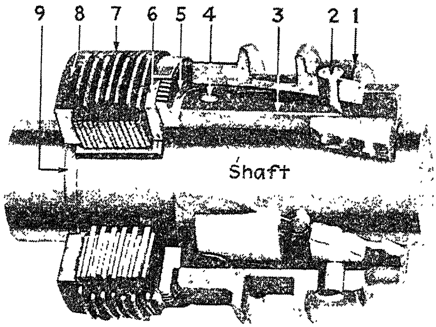


FIG. 38*B*.—The *Pullmore* multiple-disk clutch. (*Courtesy of The Rockford Drilling Machine Co.*)

The clutch yoke is not shown; it fits in the deep slot in the shipper sleeve (1). The sleeve (1) carries the dog pressure pin (2) and forces down the end or toe of the dog (3). The dog (3) operates on the dog pivot pin (4) and the heel of the dog is pressed against the cam which is part of the adjustment collar (5). The collar (5) presses against the pressure plate (6) which in turn compresses the entire disk stack. The end plate or thrust plate (7) is held from lateral movement by the split ring (8) held in a groove in the shaft.

48. Clutches.—Several kinds of clutches are used in machine tools to permit one part to engage another and yet be easily and instantly disengaged. Those most used are: the positive jaw clutch, the cone clutch, the expanding-ring clutch, and the disk clutch.

The jaw clutch, either radial or saw-tooth form, is used for positively engaging (no friction) the two members. The

clutch may be engaged or disengaged only at comparatively slow speeds. It is used, for example, in certain reverse mechanisms, and in the slower change gears.

The cone clutch is a friction clutch and is used for slow-moving parts and comparatively light duty, as for lathe-apron-gear clutches.

The expanding ring is probably the most efficient of the older forms of friction clutches. It must be comparatively large to deliver much power, and if the speed is high the wear is rapid. Examples of its use are the drill-press tapping attachment and the driving pulleys on the countershaft.

The disk clutch (Fig. 38) is the most efficient clutch yet devised. Disk clutches are made with single or multiple disks, in single or double form, in many sizes, and for operating in a bath of oil or with dry disks. The difference in construction between the oil type and the dry is in the outer disks, which in the dry type are made of bronze. In the oil type all disks are hardened steel. The multiple-disk clutch, running at high speed in a bath of oil, is the present trend in machine-tool drives. These clutches have been improved until now they take very little space yet are of great power, flexibility, reliability, and durability.

49. Attachments.—A great expense in manufacturing is the obsolescence of special machines. By building special units or attachments it is possible often to increase the production capacity of a standard machine, without destroying its flexibility, to practically the output of a special machine. If the machine is to be used in making a special part only a portion of the time, being a flexible machine tool, it may be used for other work the rest of the time. If the part produced becomes obsolete, then only the special unit is obsolete, not necessarily the whole machine.

The demand for special attachments to increase production, speed, and accuracy presents real problems for competent designers and for skilled machinists and toolmakers. It only means, however, the application of the old principles of

machine-shop practice—the application of screws, gears, cams, levers, clutches, and properly sharpened cutting tools. It calls for real mechanical ability to design and build special mechanisms, and the machine shop is where the training is obtained.

50. The Electric Motor.—It would be practically impossible to use many of the attachments, or even the improved units, of the modern machine tool, without the individual electric-motor drive, not only for the machine itself but for the attachment or unit. The speed obtained and the ease of operation are especially noticeable in the traverse of heavy units, as planer heads for example, and in the use of hand tools, as hand drills. Every young machinist should learn the principles of the construction, operation, and repair of electric motors and switches.

51. Antifriction Bearings.—The outstanding characteristics of increased power and higher speeds in the newer designs of lathes and other machine tools have been advanced by the use of "antifriction" bearings of both the ball and roller types.

The value and use of ball bearings have greatly increased with improved design, materials, and finish. One type, for example, is suitable for grinding-machine spindles having speeds up to 20,000 r.p.m., and another type for precision-lathe spindles subject to heavy duty as well as light and extremely accurate work.

Roller bearings have many new applications, even for the heaviest demands of machine spindles and shafts, markedly so since the decided development of steels and their heat treatments for the various bearing parts.

Among the advantages of antifriction bearings are: high speed when carrying radial, thrust, or combined loads; close adjustment and corresponding absence of vibration and chatter; ease of original assembly, and of adjustment when necessary; compactness of design; and accurate alignment throughout the life of the bearings.

The *load* on a bearing may be defined as the resistance to the force acting upon it. Every bearing must have the

capacity to carry its load, and, to be economical, a bearing must have the ability to carry its load at the required speed or speeds for a duration of time, say for a life of 5000 hours. Load-speed-life is the measure of the value of a given bearing.

A load at right angles to the axis of a bearing is a *radial* load, in line with the axis is a *thrust* load, and the bearings are named accordingly.

A straight roller bearing is for a radial load exclusively; straight rollers cannot be used in a thrust bearing. Taper roller bearings and ball bearings, however, may be used for thrust bearings exclusively or for radial bearings exclusively, or for any combination of radial-thrust loads. As a matter of fact all taper roller or ball bearings which are designed primarily for radial loads have also a considerable capacity for thrust loads.

By referring to *D* in Fig. 39 it will be observed that in the Timken bearing it is the angle of the cone that is largely responsible for the percentages of load ratings—thrust or radial, and that in the ball bearings it is the angular contact due to the position of the ball race in the outer ring.

By the term *preloading,* which is used so much in speaking of antifriction bearings, is meant a *specified amount of pressure* on the bearing at the time of installation or later adjustment. It is, in fact, a more or less severe tightening of the balls or rollers in their races. It is done, in machine tool spindle bearings for example, to insure rigidity under the working load, that is, to insure smoothness of operation, no vibration. If the antifriction bearings in a lathe were only fairly tight a considerable looseness would occur under the working load, but when the bearing is properly *preloaded* the additional working load does not noticeably increase the deflection.

Preloading is accomplished in the Timken bearing by tightening the cone where the cup is the fixed part (cone-adjusted bearings, *a* and *b*, sec. *E*, Fig. 39) or tightening the cup where the cone is fixed (cup-adjusted bearing, *c*, sec. *E*, Fig. 39). In the ball bearings it requires a duplex arrange-

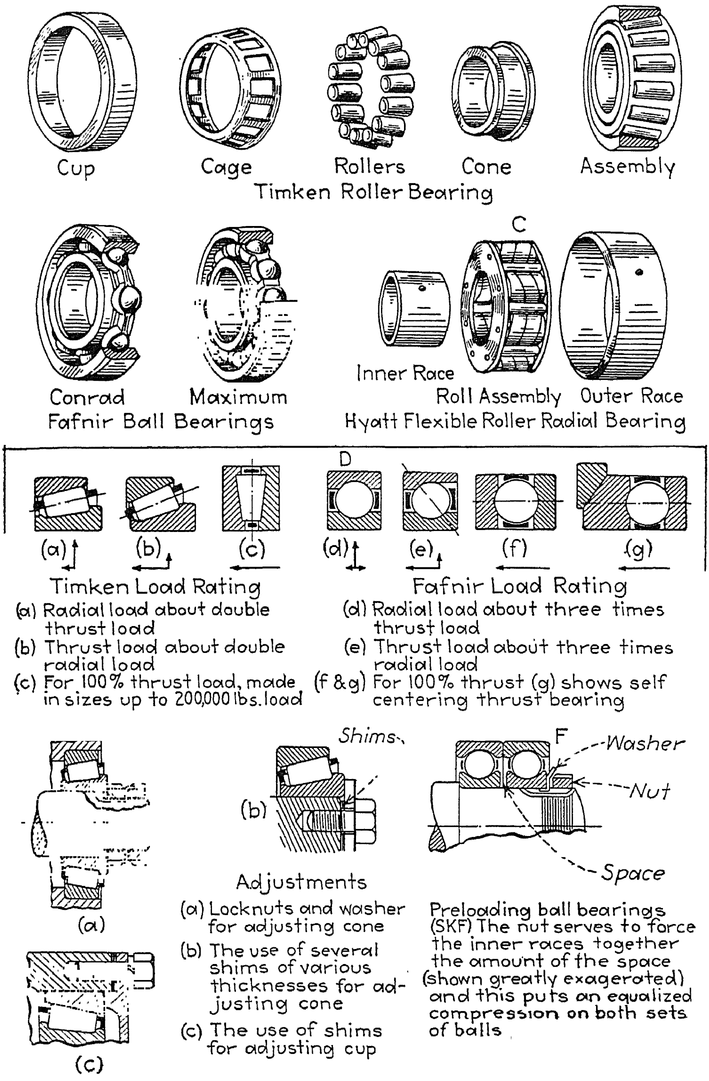


FIG. 39.—Antifriction bearings.

ment (two bearings). The inner rings are a *predetermined* number of thousandths of an inch narrower than the outer rings. The ball race is in the middle of each ring and when the bearing is installed and the inner rings are forced together the few thousandths by the locknut, the balls are tightened (preloaded) with an angular deflection in the races. This is shown exaggerated in *F*, Fig. 39.

Each type of bearing has its advantages. The Hyatt roller bearing, *C*, Fig. 39, is for radial loads only, and is especially valuable in such installations as heavy-duty pillow blocks and line shaft bearings.

The Timken taper roller bearings are available in great variety in a multitude of sizes. They are much used in machine-tool construction for gear shaft and spindle bearings for speeds up to 2000 r.p.m.

Ball bearings have the least friction and may be run at the highest speeds. Their comparative thinness is often a consideration. They may be scientifically preloaded, that is, the amount of space between the two inner rings may be calculated and provided to give the required pre-load for the given bearing, and this feature has come to be regarded as a distinct advantage.

The Needle Bushing (Torrington) which consists merely of a shell containing many small straight rolls made to a high degree of accuracy is especially useful where the maximum diameter is limited. The space required for installation is no greater than for the regular sizes of bronze or babbitt bearings. With a hardened shaft or sleeve the Torrington bearing may be used for speeds up to 5000 r.p.m.

It should be understood that to cover requirements, there are many designs of antifriction bearings, and there are many devices for adjustment, methods of lubricating, and various types of closures for keeping foreign matter from the bearings. To one particularly interested the manufacturer will furnish detailed illustrations, descriptions, and instructions.

52. Adjustments of Bearings, etc.—Even in the strongest, most modern lathe occasional adjustments are required. It

becomes necessary, from time to time, to make adjustments in any machine. Necessary adjustments should not be postponed; parts wear rapidly as soon as any "shake" is permitted. This is true for any kind of bearing—flat, plain, or antifriction. Means of adjustment are provided in all machine tools, such as gibs—straight or taper—for flat bearings; shims between the plain bearing and its cap; check nuts or shims for antifriction bearings. For any kind of bearing it takes knowledge and care to *make the proper adjustment.*

Manufacturers of machine tools provide instructions for important adjustments. Even the experienced machinist

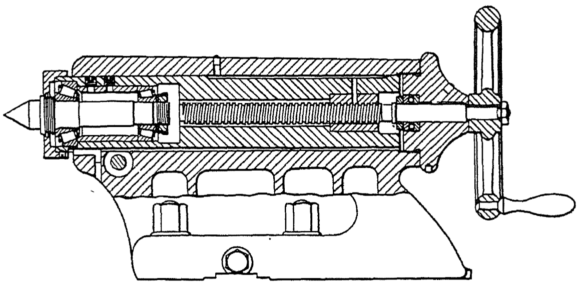


Fig. 40.—An example of a modern tailstock with built-in antifriction center. (*Courtesy of The American Tool Works Company.*)

should be sure of the construction to avoid extra work or later trouble, or both. The beginner had better get his first lessons in this important detail of the trade from watching the able mechanic.

If necessary to take a unit apart, use the wrench or screwdriver that fits; keep the parts together in a suitable box. When assembling and adjusting, have the parts clean; be careful to set up the screws and bolts tight enough, but not too tight; take care not to have too much pressure on the bearings.

53. Examples of Modern Designs.—The revolving dead center with antifriction bearings (Fig. 40) illustrates the tendency in design. Every machinist realizes the advantage of such a center in high-speed production work, provided the

work can be kept cool enough not to expand unduly, or if extreme care is taken to adjust the center if and when necessary.

A modern lathe headstock is illustrated in Figs. 41 and 42. Attention is called to the features discussed in the preceding pages, for example: multiple speeds controlled by clutch and

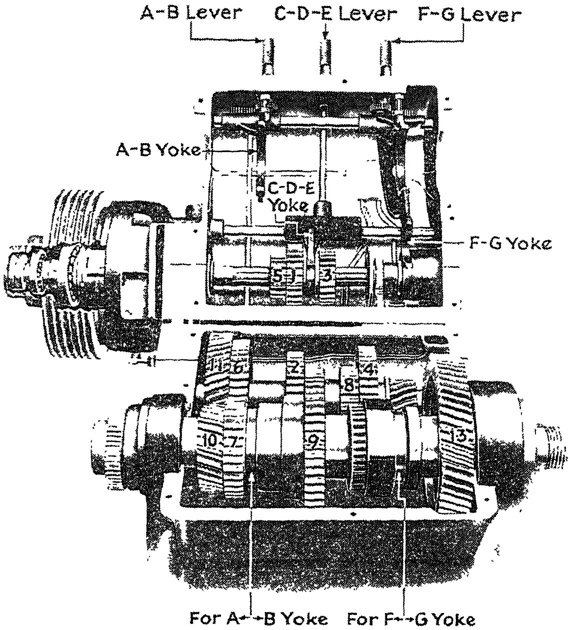


FIG. 41.—Geared head for lathe. Twelve spindle speeds are provided. For lever arrangements and gear runs see Fig. 42. (*Courtesy of The American Tool Works Company.*)

brake; automatic oiling; sliding gears on multiple-splined shafts; helical gears for smoother action; antifriction bearings; compact, rigid construction that makes for smooth running; and convenient speed-change levers.

A study of these illustrations, to solve the gear runs for example, will prove interesting. This is the kind of mental exercise that makes the expert machinist.

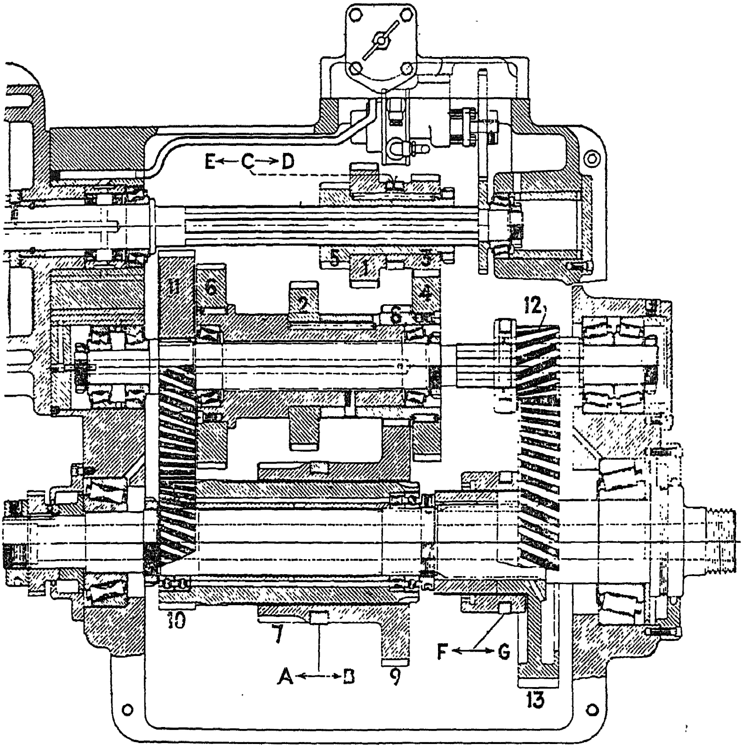


FIG. 42.—A diagram of the gears in Fig. 41.

Gear runs	Levers	Spindle speeds
1, 2, 6, 7	*A-C-F*	668
3, 4, 6, 7	*A-D-F*	576
5, 6, 7	*A-E-F*	338
1, 2, 8, 9	*B-C-F*	240
3, 4, 8, 9	*B-D-F*	172
5, 6, 8, 9	*B-E-F*	122
1, 2, 6, 7, 10, 11, 12, 13	*A-C-G*	86
3, 4, 6, 7, 10, 11, 12, 13	*A-D-G*	62
5, 6, 7, 10, 11, 12, 13	*A-E-G*	44
1, 2, 8, 9, 10, 11, 12, 13	*B-C-G*	32
3, 4, 8, 9, 10, 11, 12, 13	*B-D-G*	22
5, 6, 8, 9, 10, 11, 12, 13	*B-E-G*	16

Due to the 100 % antifriction construction of the head mechanism, these speeds may be doubled if desired.

CHAPTER III

CUTTING TOOLS AND CUTTING SPEEDS

54. Terms Used.—In the beginning of a discussion of lathe tools it is necessary to call attention to the work being done (1936) in respect to the standardization of terminology and definitions of cutting tools for lathe, shaper, planer, etc. Committees of the National Metal Trades Association, the American Society of Mechanical Engineers, and the Society of Automotive Engineers are engaged in finding terms and definitions for the shapes and elements of these tools which when agreed upon by the members of the several committees will be acceptable to the American Standards Association.

It is only necessary to give two or three examples to show existing incongruities, and when one considers the need of exact understanding and agreement in terminology relating to such an important factor in production as the cutting tool, the wonder is that this standardization has been put off so long. To begin with, as at present termed, the right-hand tool cuts to the left and the left-hand tool cuts to the right, which is illogical to say the least. The terms tool angle, cutting angle, and lip angle are taken by some to mean the same thing, by others to mean different things. True clearance, effective clearance, and angle of freedom are different terms meaning exactly the same clearance. A tool has rake, that is, it slopes, or has an inclination, from the front toward the back. Some call it front rake, others call it back rake, and it has other names too.

All in all there are many misnomers, contradictions, and disagreements. A standardization will be very welcome, especially if the terminology and definitions as finally accepted are logical, simple, brief, and few in number.

When the standardization is complete, if any changes are made in the names and definitions as given in this book it will be a simple matter to apply the new terms.

55. Cutting Tool Efficiency.—It is very necessary for the student in machine-shop practice to realize in the *beginning* that the cutting tool is a most important factor. There is nothing in shop work that should be given more thoughtful consideration than cutting tools. If one understands the principles underlying the successful action of the cutting tool, he has gone a long way in becoming expert in its use.

Time is always wasted if an improperly shaped tool is used. A dull tool is a disgrace to the shop. It is fairly difficult for the beginner to hold a lathe tool against the grinding wheel and grind it just how and where it should be ground. He must first learn how it should be ground, and he must acquire by practice the knack of grinding it.

The action of a cutting tool depends primarily on three things—(1) the rigidity of the work, that is, of the piece itself, and the manner in which it is held in the machine; (2) the rigidity of the tool—its size and the way in which it is held; and (3) the shape of the cutting tool as it is ground, and as it is presented to the work.

The machine-tool builder takes care of the design of the machine to give the necessary strength and stability. The cutting action of the tool, however, depends on its shape and its adjustment in the holding device. This is especially interesting to the machinist because most of the cutting tools he uses in the shop must be shaped—or at least sharpened—and adjusted (set) by himself. The experienced workman will never use a dull tool, a poorly shaped tool, or a tool improperly set or insecurely held if he can help it.

56. Cutting Tools Used in Lathe Work.—There are many operations that may be done in a lathe—turning, cutting a thread, boring a hole, etc., and each operation requires its own particular tool. Formerly lathe tools were forged and the chart, Fig. 43, shows several shapes of forged tools. In recent years, however, for reasons of economy and convenience, the

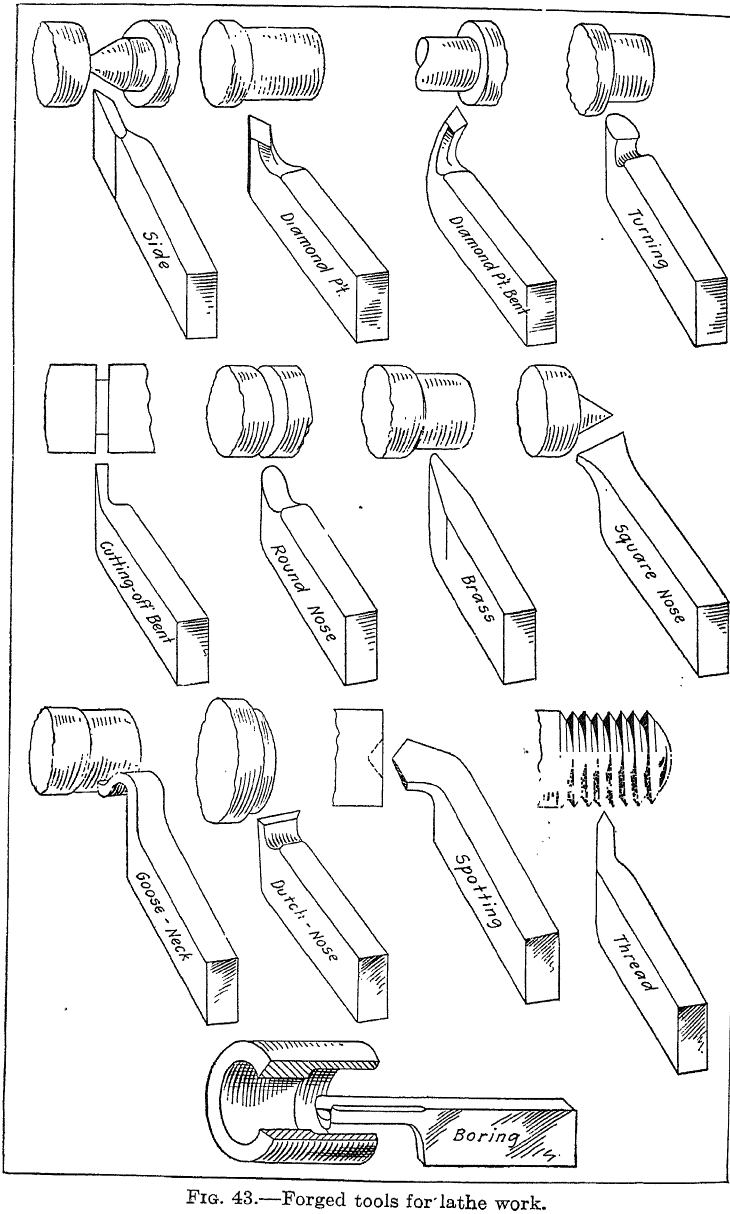


FIG. 43.—Forged tools for lathe work.

FORGED LATHE TOOLS

1. Side Tool.—Used for facing. Has side clearance and front clearance. The front is ground at an angle sufficient to clear a lathe center. It has side rake but no front rake.

2. Diamond-point Tool.—A turning tool which is practically obsolete; largely superseded by a shape suggested by Dr. F. W. Taylor.

3. Bent Diamond-point Tool.—Diamond-point tools, side tools, thread tools, cutting-off tools, and others are often bent to either the right or left as desired.

4. Turning Tool.—This is substantially the shape recommended by Dr. Taylor after years of experimenting as being the most efficient form of turning tool. Medium front rake 8 deg., medium side rake 14 deg. Sometimes called a bull-nose tool.

5. Cutting-off or Parting Tool.—Explained fully on page 166.

6. Round-nose Tool.—Made with round cutting edge of any required radius and used for filleted shoulders and also for "necking" or "grooving."

7. Tool for Turning Brass.—Ground substantially like a small round nose. Has no rake because brass is soft and a tool with rake is apt to dig.

8. Squared-nose Tool.—Very useful for truing live centers or turning short tapers. Used also with a coarse feed for a very light finish cut on cast iron. Has no top rake.

9. Goose-neck Tool or Spring Tool.—Used for turning large fillets and other forming work, when a wide cut is required, also for finishing cast iron and for water finish on steel. Any tendency of this tool to spring is away from the work.

10. Dutch-nose or Shovel-nose Tool.—A very efficient tool for facing work of a fairly large diameter when considerable stock is to be removed. It is fed toward the center. Used also for finish turning steel, coarse feed and light chip. Used with soda water for a high finish. Has front rake but no side rake. Has clearance on both sides and may be used either right or left.

11. Spotting Tool.—This tool is ground to an angle of about 120 deg. to correspond with the angle formed by the lips of a drill. Note that each lip is given clearance but in opposite directions because the cutting force is up on one lip and down on the other. Used to make a central spot in which to start the drill. Often called centering tool.

12. Standard Thread Tool.—Ground accurately to 60 deg. gauge to form a thread on a cylinder. Explained more fully in chapter on Threads and Thread Cutting.

13. Boring Tool.—Explained fully on page 187.

forged tools have been largely superseded by various forms of toolholders which will securely and rigidly hold turning and boring *tool bits,* side-tool and cutting-off-tool *blades,* etc., made of high-speed steel. This saves tying up a large amount of

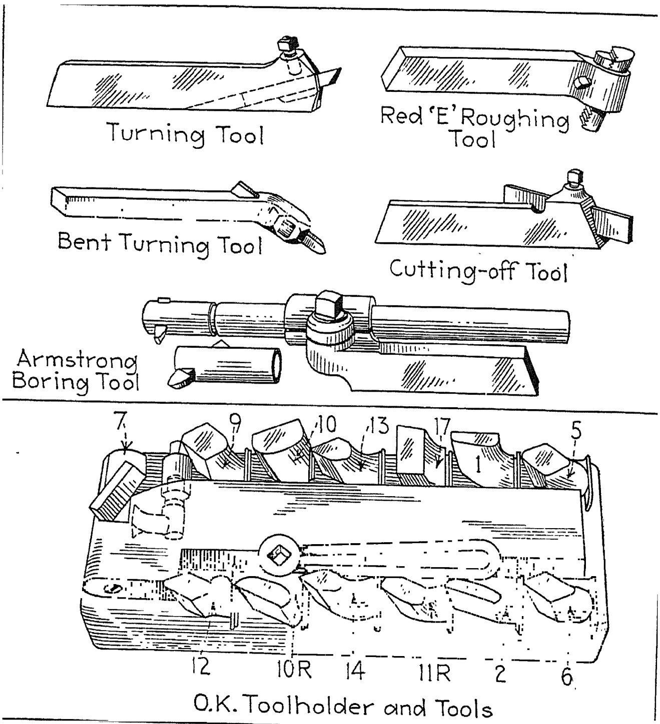


FIG. 44.—Representative types of patented toolholders.

steel and also the expense of forging. Figure 44 illustrates examples of lathe toolholders.

Some of the more common shapes for lathe work for which the regular square tool bit may be ground are illustrated in Fig. 45.

The various kinds of tools will soon become familiar to the young man in the shop; these charts, however, are instructive at this time as illustrating an excellent example of the impor-

tance of a knowledge of principles. The underlying principles of cutting angles, clearance angles, etc., which are explained in the following paragraphs, are the same for any machine-shop cutting tool. *If one knows why a turning tool is ground a certain shape, why it has certain angles of clearance, etc., and has practiced enough to hold it and manipulate it skillfully against the grinding wheel to give it this shape, he has learned* 90 *per cent of tool grinding. He will soon learn the shapes and angles of the other forms of tools, and he has acquired the skill in grinding that is fundamentally the same for all.*

The following discussion contains considerable detail, because it is so important for the beginner to know about the tool angles.

57. Steels of Which Cutting Tools Are Made.—Cutting tools used in machine shops are made of high-carbon steel (tool steel), high-speed steel, and alloys such as stellite and the cemented carbides. The stellite and carbide tools are used only in special rapid-production work. Carbon steel for cutting tools, excepting hand tools, has almost entirely given way to high-speed steel.

Cutting tools, of whatever kind of steel, are hardened and tempered.[1] Too much heat, when using the tool, will destroy the temper and make the cutting edge soft. In lathe work, or drilling, or milling—any machine operation where metal is cut—heat is generated. The chip is heated instantaneously, the work itself soon warms noticeably, and the tool point becomes very hot. In many machines a liquid coolant is used to carry away part of the heat, but in any case, if a *carbon-steel* tool is used and the speed of the cut is so high that enough heat is generated to blue the cutting edge, the temper is lost. The great value of *high-speed steel* lies in the fact that it will stand about three times as much heat as carbon steel and still retain its temper. A limit speed for carbon steel may be at least doubled when using high-speed steel; that is why "high-speed" steel is so named, and the reason for its success.

[1] For description of hardening and tempering of steel, see page 338.

58. The Cutting Edge and the Faces That Form It.—A cutting tool may have a single cutting edge, as a chisel or a lathe tool, two cutting edges as a twist drill, or several, as a

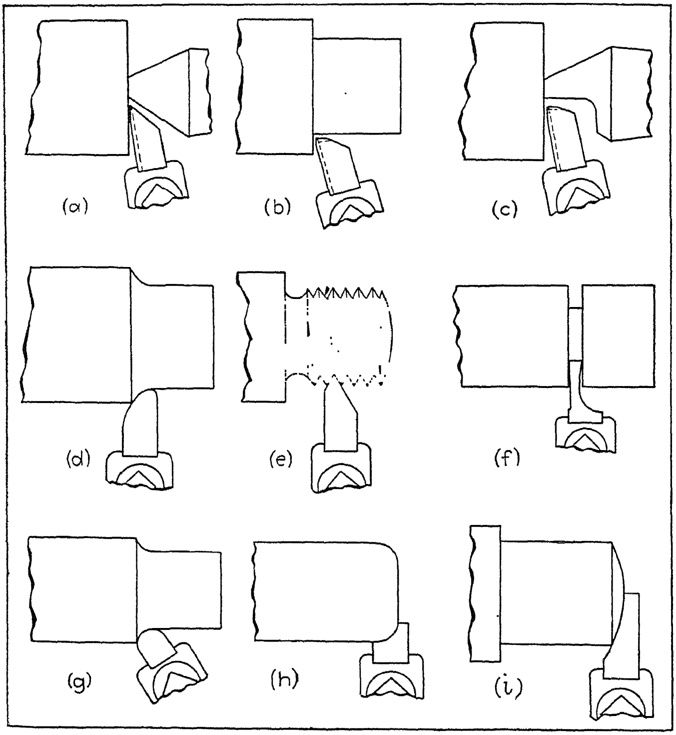


FIG. 45.—Some of the shapes to which tool bits may be ground. *a*, facing tool or side tool; *b*, shoulder tool or corner tool; *c*, shoulder tool, same as *b*, used with half center for facing; *d*, turning tool; *e*, thread tool; *f*, cutting-off or parting tool; *g*, round-nose tool (smaller sizes used for necking); *h*, corner rounding or radius tool; *i*, crowning tool. Note: Grinding a tool point on both ends of a tool is not economical.

reamer or a milling cutter. Any cutting edge is formed by the meeting of two surfaces or "faces," two plane surfaces, as for example, a flat chisel, or one plane and one curved, or both curved. Sometimes these faces are called merely the *front* or *side* or *top* as in tools for lathe, shaper, and planer. Occasionally a face is called the *lip* when referring to a lathe tool or a

drill. The front of a tooth of a reamer or a milling cutter is called the *face* of the tooth, while the other cutting-edge surface is called the *land*.

In any tool, both surfaces that form the edge must be smooth to produce a keenness of edge. If either face is not smooth, the edge is not keen, and it cannot produce a smooth cut. For example, if the groove of a milling cutter or a reamer is carelessly made, and the face of the tooth is torn and rough, no matter how smooth the land may be the edge will be as rough as the rough face. Also, a smooth edge will stay sharp longer than a ragged edge, therefore, when sharpening any tool, the finishing cut of the grinding wheel should be light, and the intelligent use of an oilstone is recommended.

When the cutting edge is broken or rounded it is dull, and to sharpen it, one of the faces, and in some tools, both faces must be ground. The grinding of a lathe tool is done, usually, by hand, moving the tool skillfully against the revolving abrasive wheel. To sharpen it skillfully means, first, to know the positions in which it must be held against the wheel to meet the requirements, and second, to gain the knack of holding it in these positions.

59. Contour of Turning Tool.—For generations past the "roughing tool" was forged and ground to have a top face substantially as shown in *A*, Fig. 46. That is, the top face was "egg shaped" with the cutting edge presented to the work at an angle of 20 to 30 deg. and the nose rounded in proportion to the size of the tool. Taylor proved that such a shape, with the round nose modified to meet conditions, will answer in most cases for roughing and finishing cuts in either steel or cast iron in the run of shop jobs in a 14- or 16-in. lathe, and for roughing cuts in the shaper, planer, and larger lathes.

In *B*, Fig. 46, is shown a shape of tool ground on a bar, which, as far as the cutting edge is concerned, is the same as the tool *A*. In *C* is shown a tool bit ground to resemble tool *A*, and in *D* the tool bit resembles the tool *B*. Practically, the four tools are alike in the shape of the cutting edges and in the cutting action.

If the round nose of a certain tool leaves feed marks too pronounced, a tool with a larger radius may be used. If in a different kind of job, the nose has too large a radius, and taking the wider chip seems to cause a chatter, it will be easy to substitute a tool with a narrower nose. Various other shapes of tools will be described from time to time. At this time refer again to Figs. 43 and 45.

60. Tool Angles.—To grind the tool properly the edge must keep its shape—flat or curved as the case may be. Also, to

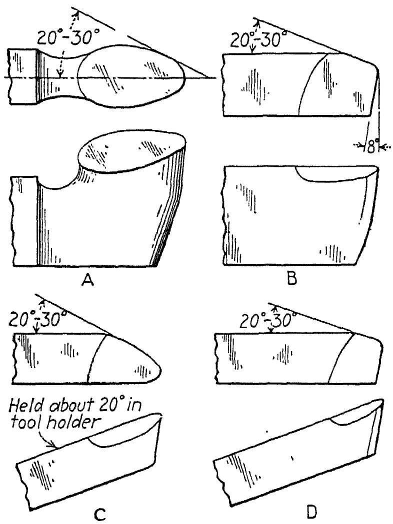


Fig. 46.—Contours of turning tools.

cut well, the surfaces that form the edge must be "ground to certain angles." These tool angles are measured in degrees and are often described as so many degrees "with the horizontal" or "with the vertical." (In this case, horizontal means in a plane parallel to the base surface of the tool, and vertical means at right angles to the base surface. Lathe tools are held in the tool post practically in a horizontal position and no doubt this is the reason for these terms.)

When reading about the tool angles, and when practice grinding, the beginner should try to have "in his mind's eye"

the value (size) of the given angle. A good way is to compare the angle to a right angle (90 deg.) in the same way one compares an eighth or a quarter of an inch with an inch. The hands of a clock set 15 min. (time) apart will measure an angle of 90 deg.; 10 min. apart will indicate 60 deg.; 1 min. apart, 6 deg. Just to make a test, draw angles of, say, 60, 45, 20 deg. and then gauge them with a protractor (see Fig. 166, page 213).

The angles of the tool are named thus: cutting angle, the angle of the wedge that cuts or peels; clearance angle, ground away so the tool will not rub; rake angle, the slope of the tool so it will peel instead of push off the chip. Discussions of these angles follow.

61. Cutting Angle.—The action of any cutting tool in any material is that of a wedge prying apart or separating the substance of the material. The angle of the wedge is the *cutting angle of the tool* (see Fig. 47).

The harder the material to be cut, the more the cutting edge must be supported, that is, the cutting angle must be greater. The cutting angle which is correct for wood is not substantial enough to stand up under the strain of cutting iron or steel; the cutting edge, not being sufficiently supported against such a severe crushing force, would soon crumble and the value of the tool would be lost. The proper cutting angle for a metal-cutting cutting tool is 60 to 80 deg. depending largely upon the hardness of the metal to be cut.

A cutting tool must be correctly shaped and sharpened, and properly set in the machine, in order to have its cutting angle effective. Many tools such as drills, milling cutters, and some lathe tools are already shaped when the machinist gets them, but he must know how to sharpen them. The tool bits, so much used for turning, boring, and planing, must be shaped, as well as sharpened and set, by the man on the machine.

Metal is ground away from the forged lathe tool—and from the tool bit—to sharpen it, and to give it after it is sharpened the shape it ought to have and, when set in the machine, the correct cutting angle. It is not enough to know that the

cutting angle for a certain tool must be 70 deg., the machinist must know how to grind and how to set the tool to get that angle. That is, when the tool is held against the grinding wheel to get the shape of a turning tool, for example, and a cutting angle of, say, about 70 deg., it is ground on the side and on the front so that it will not rub on the work. This is

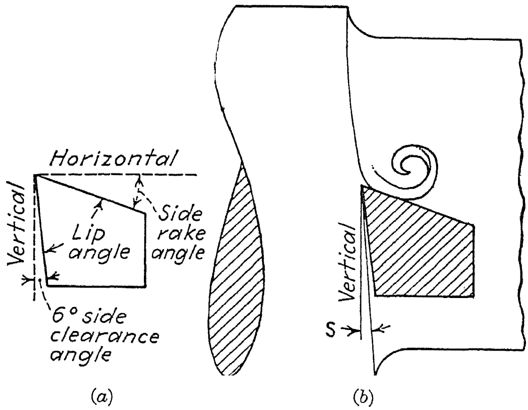


FIG. 47.—In (*a*) are illustrated the side clearance angle of 6 degrees "from the vertical," the cutting angle or lip angle of 60 to 80 degrees (depending on the material mostly, sometimes on the nature of the cut), and the side rake angle of average 12 degrees "from the horizontal." In (*b*) is shown the *action* of the tool. Note the chip being "peeled" off because the tool has rake. Note also the separation of the material just ahead of the cutting edge. Attention is called to the *slant* of the cut as indicated by the angle S, which shows that since the tool feeds (to the left in this case) it must have clearance or it will rub. The coarser the feed, and the smaller the diameter for a given feed the greater the slant, but 6 degrees side clearance is usually enough.

grinding the *clearance*, side clearance and front clearance. It is also ground (or held in the toolholder) so that the top of the tool slopes, toward the back or toward the side, or both, and this angle of slope is called *rake*. Slope from the front to the back is called front rake (or, in some shops, back rake) and the slope to the side is called side rake.

62. Clearance and Clearance Angles.—When whittling with a jackknife, the back of the blade is raised a trifle, otherwise it will merely rub instead of cut. If it is raised 10 deg. the *clearance angle* is 10 deg. In most machine-shop cutting

operations the action is similar; the cutting edge digs in, splitting or parting the metal ahead of itself, *one face* of the tool pries or peels off the chip, the *edge* tends to smooth the torn surface, and *the other face clears the work altogether.*

Three examples will illustrate the action:

1. The work in a lathe as it revolves is forced down against the cutting tool (*a*, Fig. 48), the chip is pried off against the top of the tool, and the front and side clear the work.

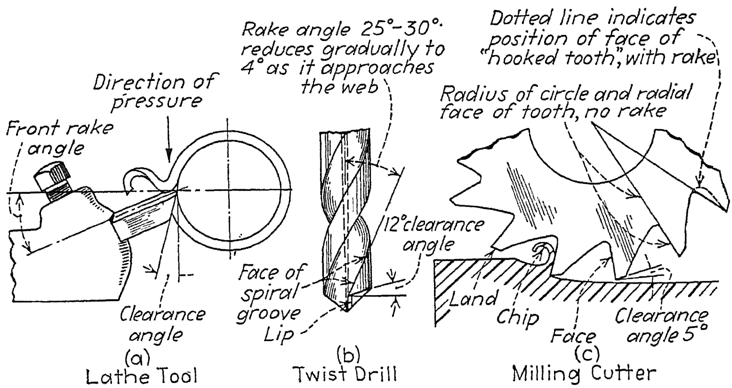


FIG. 48.—Shows clearance and rake on three much-used machine-shop cutting tools.

2. A twist drill, pressed (fed) into the work, cuts as it revolves. The cutting edges dig into the metal, the faces (of the spiral grooves) pry or peel off the chips, and the "lips" clear the work because they have been "backed off," that is, given clearance (*b*, Fig. 48).

3. In the case of a milling cutter the face of the tooth peels off the chip while the "land" just clears (*c*, Fig. 48).

In all three of the above examples one face pries or peels off the chip and the other is ground at an angle, or as the machinist says, "backed off" or "given clearance" or "relieved," so it will not rub.

In general, the angle of *side clearance* on a lathe tool should not be more than 6 deg. (Fig. 49*a*), because the greater the

amount of metal a cutting edge has under it, or the more it is backed up, the longer it will stay sharp.

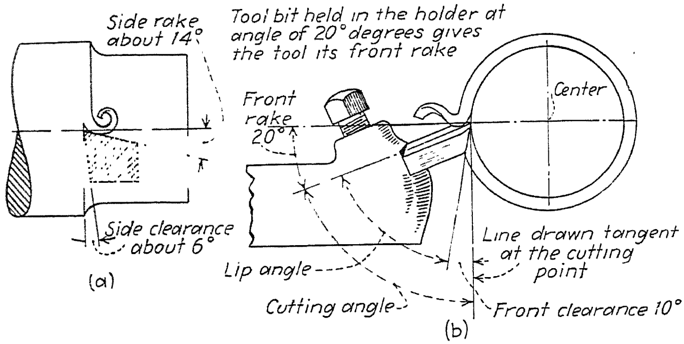


FIG. 49.—(*a*) side rake and side clearance; (*b*) front rake and front clearance. The line drawn tangent at the cutting point shows the direction of the force against the tool. Set "on center" this tool has 10 deg. front clearance. If it were set a little above center, it would have less *effective* front clearance, less cutting angle, and more rake.

The direction of the force which is exerted against the turning tool is along a line tangent to the circumference of the work at the cutting point (see Fig. 49). As the usual practice

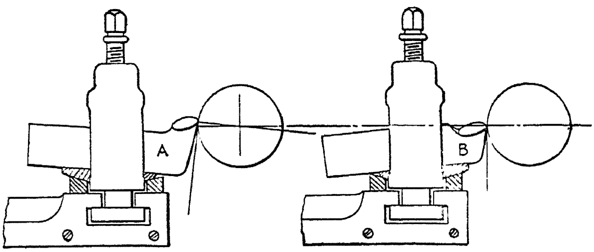


FIG. 50.—Setting a lathe tool to get the proper clearance angle. A front clearance of 10° or 12° is usually ground on a forged tool. Note the position above the center of the new tool *A* so as not to give excessive front clearance. Note *B*, which represents the position of the same tool after being sharpened several times, set on center to give correct clearance.

is to set the turning tool on center or a little above, it is necessary to have clearance at the front of the tool, so that it will not rub. This *front clearance* is usually about 10 deg. The design of the tool post (ring and rocker) permits of using

tools of various heights. Since 10 deg. is practically a standard clearance, the height of the tool will determine the amount to set the tool above center to resist the tangential force of the cut and still have sufficient clearance. This is illustrated in Fig. 50. That is, a tool may be *ground* with 10 deg. front clearance, yet when set above center have, in effect, much less than 10 deg. The amount it really has when in action in the lathe is known as *effective* clearance.

From the above it will be understood that the *effective* front clearance of a lathe tool depends upon the tool setting, the more above center the cutting edge is the less clearance it has. If set a little too high the tool will rub, and if too low the edge will be more quickly dulled.

The *effective* side clearance depends upon the amount of feed as shown by the *slant* in Fig. 47.

Note.—For many years machinists have spoken of the angle of the wedge as the cutting angle, but there has been a difference of opinion as to whether or not the cutting angle included the clearance angle. As a matter of fact the true cutting angle is the angle as ground on the tool (the *lip angle*) plus the *effective* clearance. Also it should be noted that on a tool having a curved cutting edge such as the usual turning tool, the true cutting angle is the composite of the side and end angles and is measured on the line of the chip flow.

It may help in understanding the effective clearance to imagine two other conditions than are shown in Fig. 49, possibly making a couple of sketches. In the figure the tool is ground with 60 *deg. lip angle*, is held in the holder at an angle of 20 deg., and, being set "on center" of the work, has, as shown, an angle of front clearance of 10 deg. and a cutting angle of 70 deg. Now imagine, first, that the tool is ground to have only 1 deg. front clearance and set exactly on center as shown in the figure; the lip angle would be 69 deg. and the cutting angle would remain unchanged. Second, imagine the front of the tool (as in the figure with 60 deg. lip angle) tipped up until the cutting point is just high enough to bring the tangent line to indicate 1 deg. front clearance instead of 10 deg. (*B*, Fig. 50). In this case the lip angle would remain 60 deg. but the cutting angle would be 61 deg. instead of 70 deg. as before.

63. Rake Angles.—A definition of rake is "an inclination from the vertical or horizontal." If a turning tool is set in a lathe so the top is flat and horizontal, it will have no rake,

or if a drill has straight grooves (see page 174) it has no rake. Note in Fig. 48 that both the lathe tool and the twist drill have rake, while the milling cutter, having a radial tooth, has no rake. A milling cutter with a "hooked tooth" has rake (shown in one tooth in Fig. 48). When a tool has no rake it pushes off the metal, while if it has rake it tends to peel off the chip. Most tools have rake.

Sometimes a tool is given "negative rake," for example in planer work the roughing tool is often ground with regular side rake but with negative front rake, in order to ease the blow as the tool hits the work at the start of the cut. This is shown in *b*, Fig. 55.

In lathe tools it usually is necessary to have the top of the tool on a double slope, from the front and from the side, otherwise the cutting angle will not be acute enough. Also, having the double slope causes the chip to "flow" in the desired direction. The slope from the front is called *front rake* (in some shops, back rake) and the slope from the side is called *side rake* (see Fig. 49). The number of degrees of these rake angles depends, of course, on the cutting angle required, the more rake a tool has the less cutting angle it has, that is, the sharper the wedge is. A cutting angle of 70 deg. is average for cast iron and tool steel, and 60 deg. is average for machine steel. In a toolholder the front rake is taken care of, usually, by the position of the bit at an angle in the holder (Fig. 52). The side rake may be nicked in the bit or it may be ground as illustrated in Fig. 51.

Brass is a softer material and therefore would not seem to require so heavy a cutting angle as steel. However, no rake is given a cutting tool for brass because of the tendency of the tool to "hook in" or "dig in" the soft material.

64. Keep Cutting Tools Sharp.—A cutting tool carefully ground will stay sharp under correct working conditions for a considerable time, but as soon as it is noticeably dull it should be reground or the tool and possibly the work will be ruined. A dull tool tears rather than cuts the material; it springs the work, and does not make a smooth cut. To *keep cutting*

tools sharp is a most important factor in efficient machine work.

It is an acknowledged fact that lack of judgment in the grinding of tools costs thousands of dollars every year in the wastage of materials alone. To sharpen a dull tool shall it be ground on the top, on the front, or on the side, or a little here and there? No rule can be given. Each tool grinding calls for judgment. Examine the particular tool; how many times may it be ground on the top before it is worn out?

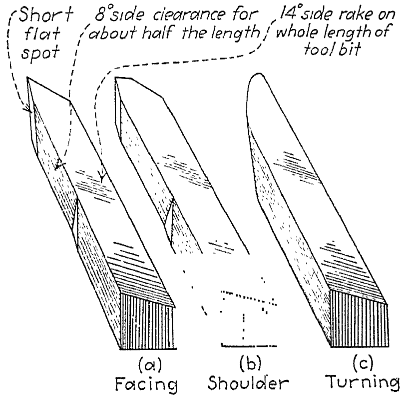


FIG. 51.—Tool bits beveled on the top for side rake angle. Sharpened by grinding on the end only.

How much may it be ground on the side before it is too thin? How much of the life of the tool is lost by grinding it on the front? Keep these things in mind when sharpening a tool.

In this connection refer again to Fig. 51; regular tool bits for the lathe, shaper, and planer are sharpened by grinding on the end only, when previously beveled as shown. The tool bit is ground in a surface grinder for a side rake of 12 to 14 deg. its whole length. It is an improvement for the usual facing, shouldering, and turning tool and for standard thread tools. (For shaper and planer bits the bevel is from the opposite edge.) Side rake is thus provided once and for all. All sharpening is done on the end and the bits do not

have the usual ugly nicked points. The economy in grinding time and the saving of the tools are notable.

65. Grinding Cutting Tools.—The beginner should grind a practice piece of machine steel (preferably a little larger than the tool bit so as not to get them mixed) and acquire the knack before attempting to grind an expensive tool bit. It may be fairly difficult at the start to grind the proper front clearance because the bit when in use is held at an angle in the holder. Until the eye is trained, use a gauge. The 60-deg. center gauge is suitable, if 10 deg. front clearance is wanted, because in most of the holders the tool bit is set at an angle of 20 deg. with the horizontal (Fig. 52). For any other than 60 deg.

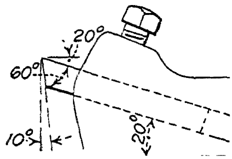


FIG. 52.—Tool bit is held in the holder at an angle of 20 deg. to give the front rake.

cutting angle it is easy to cut out a small sheet-metal gauge of the angle desired.

When grinding carbon steel, care must be taken not to bear on too hard or the edge will become blue and the temper lost. A wet grinder should be used if one is available. It is not so easy to burn the temper out of a high-speed cutter but it is easy to cause surface cracks by not having water enough. Have plenty of water and do not bear on too hard; give the wheel a chance.

A tool bit should *not* be ground in a holder, first, because it is clumsy and inefficient, and second, because one is liable to grind the holder. If occasionally the holder is ground a little, soon it is ruined Fig. 53 shows the correct way to hold the turning-tool bit. Hold it securely but not rigidly. As it is swung from *a* through *b* around to *c* (Fig. 54) it pivots slowly between the left thumb and forefinger, with the pressure mostly with the right forefinger. Grind one continuous cut,

keeping in mind the front and side clearance. Tip it a little to the right, as at *a*, to grind the side clearance, and to the left as it reaches *c* in order to finish the round nose.

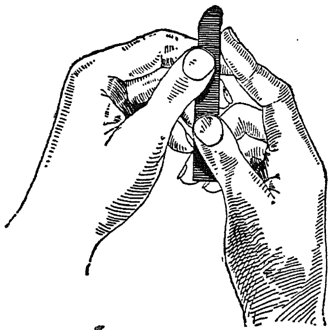


FIG. 53

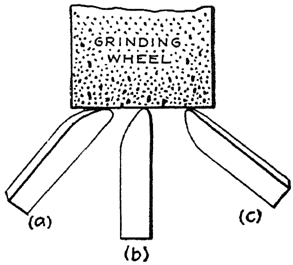


FIG. 54.

Oilstoning the edge serves to produce a better finish on the work, and prolongs the life of the tool.

DON'TS IN TOOL GRINDING

1. Don't grind stupidly; know where and why and how to take off the metal.
2. Don't hold the tool as if your fingers were paralyzed, hold it securely enough to control it.
3. Don't whittle the tool, grind it.
4. Don't hold the tool left handed or otherwise awkwardly; hold it properly, it's the easiest way.
5. Don't be afraid to use plenty of water.
6. Don't hold the tool in one place or you will cut a groove in the wheel.
7. Don't use a wheel that is grooved or out of round if you can help it.
8. Don't grind on the side of a wheel except when necessary. When it is necessary you will want a flat surface and it won't be flat if you or anyone else has cut grooves in it.
9. Don't hold the smaller tools on the tool rest, support them in the left hand and rest this hand.

10. Don't use the tool rest with more than $\frac{1}{16}$-in. space between it and the wheel.

11. Don't make a round nose of a thread tool or a thread tool of a round nose; it is wasteful of material and in the end wasteful of time.

12. Don't use the grinder without some sort of eye protection—goggles or guard.

66. Cemented Carbide Tools.—A word should be said here about the cemented carbides, the wonderful super-speed cutting materials. First, what they are: Pure tungsten, carburized to form tungsten carbide, was produced about the end of the last century by Henri Moissan. It is one of the hardest known substances, but is brittle and porous. It was not commercially valuable until a method of *cementing* it (holding tiny particles together with a suitable binder) was developed (about 1927). Since then it has been found that carbides of tantalum, titanium, molybdenum, and several others are valuable, and mixtures of different carbides that will give various *grades* of product, each superior for its particular class of work, have been developed.

The carbide, or a mixture of two or more carbides, and the binder (cobalt is much used) are powdered and mixed in the most thorough way, and then molded under hydraulic pressure into the shapes desired. These ingots or rough shapes are next pre-sintered at about 1500°F. and have then the consistency of the graphite in a pencil. In this condition they may be further shaped by turning, drilling, filing, etc., somewhat oversize to allow for later shrinkage. The pieces are final-sintered in special furnaces at temperatures ranging from 2500 to 2900°F. depending upon the grade. During this process the binder coalesces and cements the carbide particles together, forming a structure of extremely hard carbide crystals in a tough binder.

The cemented carbide thus produced has extraordinary hardness, high compression strength, low heat conductivity, but is quite brittle. Due to its brittleness, it cannot be used as a tool bit in a holder, like a high-speed-steel bit for example,

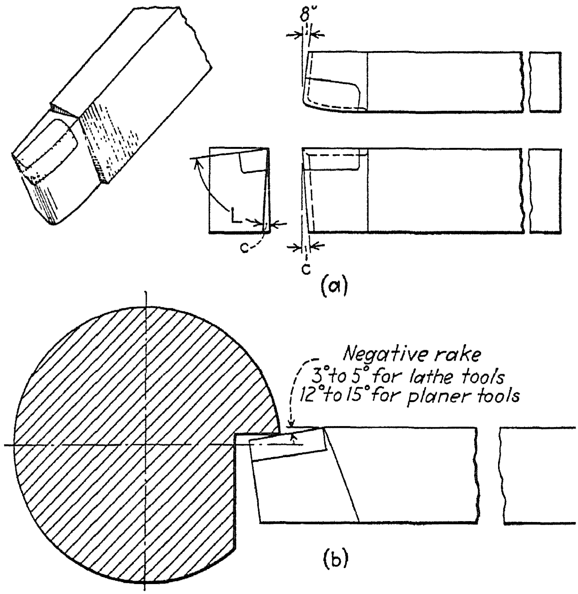


FIG. 55.—Cemented carbide-tipped tools. *a*, views of turning tool, angles marked *C* are clearance angles and *L* is the lip angle; *b*, shows how an interrupted cut in lathe work (or the beginning of a cut in shaper or planer) strikes the tool having negative rake; the impact is some distance behind the nose and is less likely to break the tip.

TABLE ANGLES FOR CARBIDE-TIPPED TOOLS

Material to be machined	Clearance angles, *C*	Lip angle, *L*
Soft gray cast iron	5°	74°–80°
Hard gray cast iron	4°	74°–80°
Chilled cast iron (65–90 scleroscope)	3°	82°–86°
Soft steel	6°	60°–65°
Hard steel	5°	65°–74°
12% manganese steel	4°	80°–84°
Stainless steel	5°	65°–74°
Soft steel castings	5°	68°–73°
Hard steel castings	5°	73°–78°
Bronze, brass, etc.	6°	65°–75°
Aluminum alloys	8°	50°–55°
Planer Tools	As above but with negative back rake 12°–15°	

but is fitted into and brazed in place to be used as the cutting point in the end of a bar of steel (Fig. 55) or a reamer, drill, or other cutting tool. Because of its low conductivity of heat it never becomes hot enough to melt the brazing material.

Cemented carbides have qualities that make possible cutting speeds several times as fast as high-speed steel. Also they will freely cut hard substances that steel will not cut at all. Their first cost is high, but there is no doubt of their value in production work, not only as cutting tools, but for parts that must be wear-resisting such as guides, gauges, wire dies, etc.

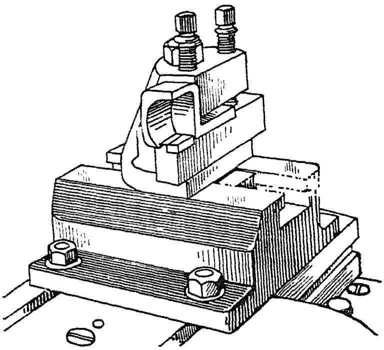


FIG. 56.—Toolholder and patented chip breaker. The chip is tightly curled, broken in 3- or 4-in. lengths and directed into the lathe chip pan. (*The American Tool Works Company.*)

It should be emphasized, however, that particular *grades* are made for given purposes, and *cemented-carbide tools cannot be used indiscriminately on various materials.* Nor can cemented-carbide cutting tools give satisfactory results unless the greatest care is taken to eliminate all vibration in work holder and toolholder. Further, safety demands protection against the tendency of the red-hot chips to fly. Note the substantial toolholder and chip breaker illustrated in Fig. 56.

Catalogues furnished by the manufacturers give details of selection, grinding, and use, and are interesting and instructive, but expert advice, in the selection of the suitable grade, is always recommended. When necessary to sharpen the tool, knowledge, care, and skill are needed.

Questions on Cutting Tools

1. What is the general shape of the cutting edge of a turning tool for lathe work? What are the disadvantages of a sharp point on a turning tool?

2. What is meant by cutting angle? How many degrees are included in the cutting angle of an average turning tool? Why not 90 deg.? Why not 30 deg.?

3. What is meant by clearance angle? How much side clearance has a lathe turning tool? How much front clearance? What is the disadvantage of too great a clearance angle?

4. What is meant by front rake? Side rake? What is the object in giving rake to a tool?

5. Other things being equal, will a tool with side rake cut equally well if fed in either direction?

6. What is meant by negative rake?

7. What is a right-hand turning tool?

8. What is a bent tool?

9. It may be stated that a cutting-off tool has five clearance angles; where are they?

10. It may be stated that a side tool has four clearance angles; where are they?

11. When it is necessary to sharpen a side tool, is it ground on the top, on the side, or on the front? Why must judgment be exercised?

12. What are the advantages of a tool that has been oilstoned?

13. If a tool rubs on the work, what faults may be found?

14. What is the chief advantage of a toolholder and bit?

15. Should the bit be ground in the holder or should it be removed before grinding? Give reasons.

16. Name three things on which the action of a cutting tool depends.

17. What is meant by the *contour* of a tool face?

18. What is meant by *effective* clearance?

19. What is the difference between the corner-rounding tool and the crowning tool?

20. What is the purpose of beveling the top surface of a tool bit?

21. What is meant by the term *cemented* in referring to carbide tools?

22. What is the difference between a tool bit and a tipped-tool?

23. What is the meaning of the word *sintered?*

24. What do you understand by the words tungsten, tantalum, and molybdenum?

25. How is the carbide tip fastened in place?

SPEED, FEED, AND DEPTH OF CUT

67. Roughing and Finishing Cuts.—There are two kinds of cuts in machine-shop work called, respectively, the "roughing cut" and the "finishing cut." When a piece is "roughed out," it is fairly near the shape and size required, but enough metal has been left on the surface to finish smooth and to exact size.

Bars of steel, forgings, castings, etc., are obtained, if possible, of the shape and size that will machine to the greatest advantage, that is, usually with one roughing and one finishing cut. Sometimes however, certain portions of a piece may require more than one roughing cut. Also, in some jobs, for example, when great accuracy is not needed, or when a comparatively small amount of metal must be removed, a finishing cut may be all that is required.

The *roughing cut,* to remove the greater part of the excess material, should be reasonably heavy, that is, all the machine, or cutting tool, or work, or all three, will stand. One need not worry much about the machine being overworked, and it will soon be quite easy to judge the capabilities of the given tool. The machinist's purpose is to remove the excess stock as fast as he can without leaving a surface too torn and rough, without bending the piece if it is slender, and without spoiling the centers.

The *finishing cut,* to make the work smooth and accurate, is a finer cut. The emphasis here is refinement—very sharp tool, comparatively little metal removed, and a higher degree of accuracy in measurement.

Whether roughing or finishing, the machinist must set the lathe for the given job. He must consider the size and shape of the work and the kind of material, also the kind of tool used and the nature of the cut to be made, then he proceeds to set the lathe for the correct speed and feed and to set the tool to take the depth of cut desired.

68. Definitions.—*Cutting speed* in any machine-shop operation is expressed in *feet per minute.* In lathe work it is the number of feet measured on the circumference of the work

that passes the cutting edge of the tool in 1 min. If it were possible to measure the exact length of the chip removed in 1 min. it would measure the cutting speed in feet per minute.

The *feed* in lathe work is the amount the tool advances for each revolution of the work. For example, in turning a cylinder with 1/32-in. feed it will require 32 revolutions of the work to move the carriage 1 in. The machinist speaks of "coarse feed" and "fine feed." These terms mean nothing except when applied to lathes of practically the same size. What might be regarded as fine feed on a large lathe would be a coarse feed on a small lathe.

By the term *cut* in lathe work is meant the *depth of cut.* Suppose a cylinder of machine steel 2 in. in diameter is put in a lathe and a cut made reducing the diameter to 1 7/8 in. Regardless of the speed or feed, the depth of the cut is 1/16 in. It should now be clear what the foreman means when he says "Give it a higher speed," "Try a coarser feed," or "Take a deeper cut."

69. The Time Element.—One of the most important problems entering into machine-shop work is the time element. The time it takes to produce a finished piece of work depends largely on the rate at which the metal is removed from the original stock. The rate at which the metal is cut off depends on three things, namely, the depth of cut, the feed, and the cutting speed. Take for example the turning operation.

1. It is obvious that the cutting edge of the tool takes a deeper cut if it reduces the diameter 1/4 in. than if it reduces it only 1/8 in. It will be folly to take two cuts if 1/4-in. reduction in roughing size is necessary. One factor then is the depth of cut.

2. If every time the work revolves the tool is fed 1/64 in., it will remove a chip only half as thick as if it were fed 1/32 in. If practicable to set the feed for 1/32 in. why not get the piece turned in half the time? Another factor then is the amount of feed.

3. If this work is 2 in. in diameter and revolves 70 times in 1 min, a point on the circumference will travel about 30 ft.

in 1 min. If the cutting tool will stand 60 ft. per minute cutting speed it will not be efficient to turn at half this speed. The third factor then is the cutting speed.

There is a new problem of cutting speed, feed, and depth of cut for every job on every machine in the shop. After awhile the workman becomes expert enough to attend to these things automatically. At the start, however, these problems require close attention and certain calculations.

70. Cutting Feeds and Speeds.—There are so many conditions that determine the proper depth of cut and feed that it is impossible to give any set rule for either. The shape of the tool, the way in which it is held, the kind of steel from which it is made are factors; also the kind of material being cut, whether machine steel or tool steel, brass or cast iron; the shape of the piece being cut, whether it is rigid or inclined to spring; the nature of the cut, whether it is roughing or finishing, are all factors which must be taken into consideration when obtaining an efficient depth of cut or amount of feed.

Conditions also govern the rate at which the tool will cut, and no table can be given that will apply in all cases. Fortunately however, there are certain well-established *average* cutting speeds for various metals.

AVERAGE CUTTING SPEEDS WITH TOOLS OF HIGH-SPEED STEEL

Stainless steel and Monel metal	50 ft. per minute
Annealed tool steel	60 ft. per minute
Machine steel, wrought iron, and cast iron	80 ft. per minute
Brass	200 ft. per minute
Aluminum	300 ft. per minute

Average cutting speed with tools of carbon steel is about half the above.

Cutting speeds must not be confused with revolutions per minute (r.p.m.). A piece 2 in. in diameter will have to make five times as many r.p.m. as a piece 10 in. in diameter to give the same cutting speed. In other words, each different diameter must have a different number of r.p.m. to give the same cutting speed. If the beginner will calculate for the first few jobs the r.p.m. necessary to give the required cutting speeds,

after awhile he will become so accustomed to seeing the machine work properly that he will be able to set up without calculations and almost without thought.

71. Cutting-speed Calculations.—Cutting speed (excepting the shaper and planer) is the rate at which a point on the *circumference* travels. In the case of a lathe it is the circumference of the work; in the case of a milling machine or drill press it is the circumference of the milling cutter or of the drill. And remember, in machine-shop practice, when speaking of sizes, the *diameter* is expressed, not the circumference. Also these diameters are given in *inches*, while cutting speed is expressed in feet. To find the circumference of a piece of work (or of a drill or milling cutter) multiply the diameter by 3.14 and to reduce to feet divide by 12. However, instead of multiplying the diameter by 3.14 and dividing by 12 in every problem it is much quicker to multiply the diameter by 0.26. [The diameter multiplied by 3.14 and this divided by 12 is equal to 0.26 times the diameter. $\frac{(\text{dia.} \times 3.14}{12} = 0.26 \times \text{dia.})$] That is, the circumference in feet is always equal to 0.26 times the diameter in inches.

Further, if one had a job that figured 2 ft. in circumference it would take 20 r.p.m. to give a cutting speed of 40 ft. per minute; if the job figured ½ ft. in circumference it would take 80 r.p.m. to give a cutting speed of 40 ft. In both cases the number of r.p.m. is equal to the cutting speed divided by the circumference in feet. From these examples the following may be deduced: To obtain the r.p.m. necessary to give any required cutting speed, multiply the diameter (in inches) by 0.26 and divide the cutting speed by this product.

Since 0.26 is so nearly ¼, it may be stated that for all practical purposes the number of r.p.m. may be calculated by the following:

Rule.—To obtain the number of r.p.m. necessary to give any required cutting speed divide ¼ of the diameter into the cutting speed.

Example.—A piece of machine steel $2\frac{1}{4}$ in. in diameter is to be turned in a lathe. What number of r.p.m. is necessary to give a cutting speed of 70 ft. per minute?

Solution: $\frac{2.25}{4} = 0.56, \quad 70 \div 0.56 = 125$ r.p.m. *Ans.*

72. Value of High Speed.—In a previous paragraph (page 85) it was stated that the edge of the tool parted the metal "ahead of itself," and it might have been said further that the metal has a tendency to "pile up" on the nose of the tool, tearing the turned surface similarly as with a dull tool.

Every machinist knows that this tendency is less at the faster speeds, but the cemented-carbide cutting tools have made it possible to prove that above a "critical speed" (over 250 ft. per minute for most steels) this piled-up edge does not form, nor is there a parting ahead of the metal, and the chip and turned surface are not torn. It is under the same principle that a bullet from a rifle will pierce a pane of glass, and thrown by hand will shatter the glass.

During a recent test, using a multi-production lathe (The American Tool Works Company, Cincinnati, Ohio) with a cemented-carbide cutting tool, bars of steel were successfully turned at 350 ft. per minute. The chip breaker (Fig. 56) curled and disposed of a $\frac{3}{8}$-in. wide chip which was "as smooth as glass," and the turned surface was clean cut. The same kind of steel was being turned with a high-speed steel tool in a near-by lathe at 80 ft. per minute, with much inferior results.

Of course cemented-carbide tools are expensive, the faster lathe is up to the minute in accurate rigid construction, an intelligent machinist is employed, but nevertheless, the facts speak for themselves, and show the trend in production.

Questions on Cutting Feeds and Speeds

1. What do you understand by "time element" in machining a piece of work?

2. Name four things that may determine the proper feed for turning.

3. What is the difference between cutting speed and revolutions per minute?

4. How many r.p.m. are necessary to turn a piece of work 1½ in. in diameter at a speed of 30 ft. per minute?

5. What number of r.p.m. is necessary to give a cutting speed of 40 ft. per minute on work 2¼ in. in diameter?

6. What rule is used to find the r.p.m. of a drill to give a required cutting speed?

7. Is this same rule used to obtain the r.p.m. of work in a lathe to give the required cutting speed?

8. In turning a cast-iron pulley 12 in. in diameter, how many r.p.m. will be necessary to give a cutting speed of 40 ft. per minute?

9. Why is it that machine steel can be turned at a higher speed than tool steel?

10. If the r.p.m. and the diameter of the work are known how may the *cutting speed* be found?

11. In cutting-speed calculations the constant 0.26 D (or ¼ D) is used. How is the constant obtained?

12. What is the property that gives high-speed steel its value?

13. State two things the chip breaker does besides *break* the chip.

14. Name three things to emphasize in considering the finishing cut.

15. What is the difference between *cutting speed* and *revolutions per minute?*

CHAPTER IV

THE SCALE, CALIPER, SNAP GAUGE, AND MICROMETER

73. Accuracy.—The student in machine work should begin at once to understand accuracy in its relative terms. He should appreciate from the start the value of the various measuring tools in obtaining the degree of accuracy the given operation demands. For example: If caliper measurement is as easy, as quick, and accurate enough, then the use of a micrometer would be poor practice. On the other hand, if it is easier, quicker, and better to use a micrometer to get the measurement, it would be wrong to use the caliper. There is no more reason why a boy should not use a micrometer the first week in the shop than there is to suppose he must chip castings for six months before he can run a lathe.

74. Scales.—Among the most useful tools in the machine shop are the steel rules of "scales." They are made in a variety of kinds as "spring tempered," "flexible," "narrow," etc., and in lengths from 1 to 48 in., the most popular being the spring-tempered 6-in. scale. Most steel rules are graduated, that is, marked by fine lines upon each edge of both sides, and often at the end, in different subdivisions of an inch. The different graduations are classified by number; for example: the No. 7 graduation, which is perhaps the most used has 64ths, 32ds, 16ths, and 100ths on the four edges respectively. The No. 4 graduation, which many prefer, has 64ths, 32ds, 16ths, and 8ths. The "flexible" scale, which is very popular with machinists and toolmakers, is graduated on three edges, in 64ths, 32ds, and 100ths.

It will be observed (Fig. 57) that the graduation lines on the scale are of different lengths, the 64th lines (not shown) are the

shortest and the lines marking 32ds and 16ths and so on are successively longer, the inch line being the longest. This is for the purpose of quickly reading the measurement.

When reading a scale measurement find the nearest "significant" graduation whether eighths, quarters, halves, and of course whole inches and add or subtract as the case may be, to obtain the fraction of the finer graduation. To illustrate:

$3\frac{9}{16}$ in. is $3\frac{1}{2}$ in. plus $\frac{1}{16}$ in.; $4\frac{15}{16}$ in. is 5 in. minus $\frac{1}{16}$ in.; $6\frac{9}{32}$ in. is $6\frac{1}{4}$ in. plus $\frac{1}{32}$ in.; $2\frac{15}{32}$ in. is $2\frac{1}{2}$ in. minus $\frac{1}{32}$ in.

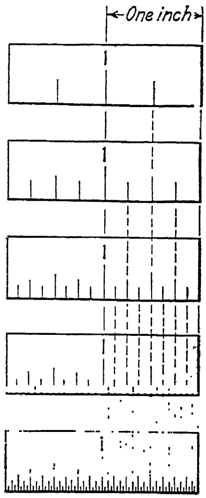


Fig. 57.

A machinist never says "two-eighths" or "four-sixteenths" or "six thirty-seconds" but such expressions as "a thirty-second over half" or "one sixty-fourth under three-quarters," etc. are commonly used. Also the terms "a scant sixteenth" or a "full thirty-second" and "half a sixty-fourth" are used.

75. Calipers.—A caliper is a tool used for measuring work. Calipers are made in several styles such as "spring calipers," "firm-joint calipers," "transfer calipers," for both inside and outside measurement.

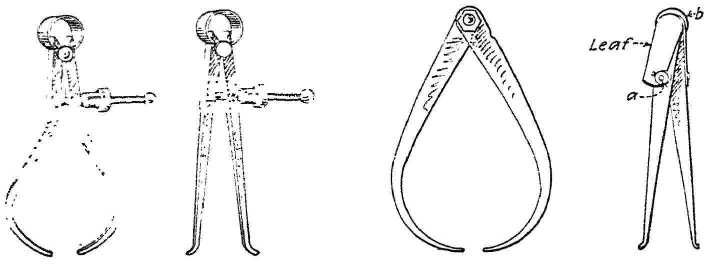
Fig. 58*a* shows "outside" and "inside" spring calipers. The spring at the top tends to keep the legs set taut against the pressure of the adjusting nut.

Fig. 58*b* shows a firm-joint caliper. A caliper of this kind is usually preferred only in the larger sizes.

Fig. 58*c* shows a "transfer caliper." The special feature of the transfer caliper is that it may be used inside of chambered cavities, over flanges, etc., removed and reset without losing the size calipered. This is done by loosening the lock nut *a* which binds one leg to the auxiliary leaf and moving this leg (while the joint *b* is tight) to clear the obstruction, then moving

it back against the stop on the leaf where it will show the exact size measured.

The caliper may be used as a measuring tool or as a gauge. The machinist generally uses it as a *measuring tool;* he takes a



(a) (b) (c)

Fig. 58.

cut for a short distance, measures it with the caliper, reads the measurement on the scale to see how much more of a chip he has to take, and proceeds accordingly. When the caliper is used to gauge work, pieces already turned for example, it is first set to the size required.

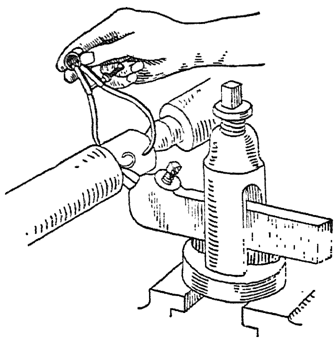


Fig. 59.

76. Measuring with a Caliper (Fig. 59).—Accuracy in caliper measurement depends on the sense of touch and to get a delicate touch of the caliper on the work it should be held lightly and not with a "grab grip." It is not difficult to demonstrate the "feel" of 1/1000 in. with a caliper. A lathe mandrel[1] tapers about 1/1000 in. in 2 in. of its length. Get a

[1] *Mandrel.*—Sometimes called "arbor," a machine-shop tool, with accurate centers, which may be forced into the hole in the piece to be machined (such as a pulley or gear), thus providing centers on which turning or other machine work may be done. For complete description of mandrel see page 152.

mandrel from the toolroom and carefully set the caliper as if to measure this mandrel 3 in. from one end, then without changing the setting, caliper the mandrel 5 in. from the end and note the difference in the feel.

If sufficient force is exerted a caliper set to 1 in. may be pushed over a piece $1\frac{1}{8}$ in. in diameter. On the other hand a caliper set to $1\frac{1}{8}$ in. can be made to "just touch' a diameter of 1 in. if held canted as shown in Fig. 60. Hold the caliper exactly at right angles to the axis of the work. *Never caliper the work while it is running;* it is not accurate and the caliper may get caught and broken.

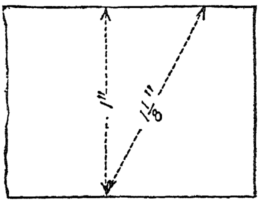
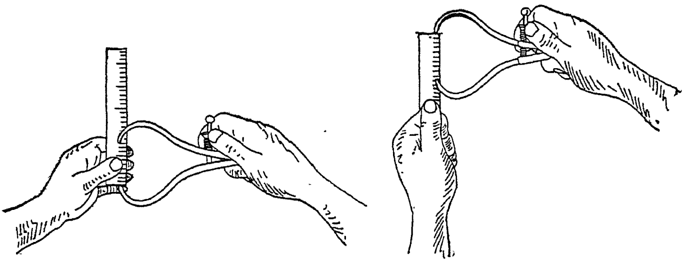


FIG. 60.

77. Setting and Reading an Outside Caliper.—To set an outside caliper hold the scale in the left hand with the end against the little finger as shown (*a*, Fig. 61) and in such a



(*a*) Correct. (*b*) Incorrect.

FIG. 61.

position that the light falls directly on the scale. Hold the caliper in the right hand, in such a position that it may be adjusted by the adjusting nut between the thumb and finger. Place the end of one leg of the caliper against the end of the scale and against the finger so that it will not slip around, and then adjust the other leg to the desired graduation on the scale. Hold the caliper true and looking squarely at the end to be set to the line, adjust the caliper until that end seems

to split the line. In this way a caliper may easily be set to within 0.002 or 0.003 in. of the exact size required. A firm-joint caliper is held in about the same way but must be adjusted by rapping lightly against some solid object.

If a plug gauge of the desired size is available or if a piece is to be duplicated the caliper may be set exactly by adjusting it to the given gauge or piece.

To read an outside caliper it is held substantially as above except that as it is not to be adjusted, the adjusting screw should not be touched.

Calipers are not *efficient* for accurate measurements but they are efficient for measuring stock, roughing cuts, lengths, and

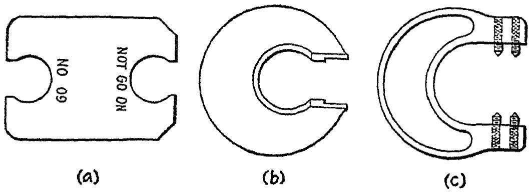


FIG. 62.—Types of limit snap gauges.

any dimensions that need not be extremely accurate. The caliper may be used if necessary for very close measurements but it is easier and quicker and surer to use a micrometer or a gauge.

In rapid-production work it is usually advisable to use gauges for determining the correct sizes. For cylindrical work the snap gauge is used and where a slight variation over or under nominal size is allowable a "limit gauge" is used.

Fig 62 illustrates three styles of limit snap gauges. It will be understood that for rough turning, the difference between the "go on" dimension and the "not go on" dimension is much greater than could be allowed in the finishing operation, and further, in different classes of work the limits allowed for finishing vary greatly. The advantage of the gauge shown at (*c*) lies in the fact that it is adjustable.

78. The Micrometer Caliper.—Micrometer calipers form convenient and accurate instruments for fine external measure-

ments. They are made in different sizes and styles to measure all dimensions to 24 in. They are graduated to read to thousandths of an inch, and one-half and one-quarter thousandths are readily estimated. Some of the calipers have verniers by which readings can be obtained to ten thousandths (see Appendix, page 380).

The essential parts of the micrometer are: *frame F* (Fig. 63) to which are fastened the *anvil A* and the *hub H*, the *spindle S* which is threaded $\frac{1}{40}$-in. pitch about half the length and which is fastened securely to the outer end of the *thimble T* and revolves freely in a fixed nut within the hub. By turning

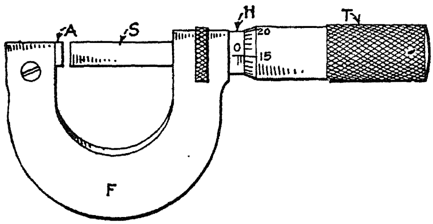


FIG. 63.

the thimble right or left one full turn, the spindle advances toward or recedes from the anvil $\frac{1}{40}$ or 0.025 in.

A line is drawn on the front of the hub parallel to the axis of the spindle and this line is divided into 40 equal parts per inch, every *fourth* part being numbered 0, 1, 2, 3, etc.

The beveled edge of the thimble is divided into 25 equal parts, every *fifth* part being numbered 0, 5, 10, etc.

When the micrometer is closed the beveled edge or the thimble coincides with the zero line on the hub, and the zero line on the bevel coincides with the horizontal on the hub. Open the micrometer one full turn or until the 0 on the bevel again coincides with the horizontal line on the hub and the micrometer is opened 0.025 in.

Open the micrometer three more full revolutions. The beveled edge of the thimble splits the 1 line on the hub and the opening is $\frac{4}{40}$ or $\frac{1}{10}$ or as usually read, "one hundred thousandths."

To read the micrometer count the numbers 1, 2, 3, etc., as 0.100, 0.200, 0.300; add each line beyond the figures as 0.025, 0.050, 0.075; then add the number of thousandths indicated

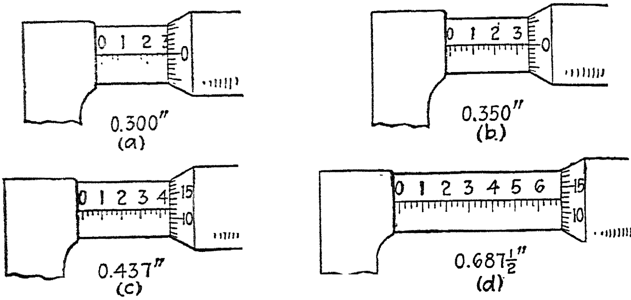


FIG. 64.—Micrometer readings.

on the thimble. To illustrate: Fig. 64, *a* reads 0.300 in. even, *b* reads 0.350 in. (0.300 + 0.025 + 0.025), *c* reads 0.437 in. (0.400 + 0.025 + 0.010 + 0.002), *d* reads, 0.687½ in. (0.600 + 0.075 + 0.012½).

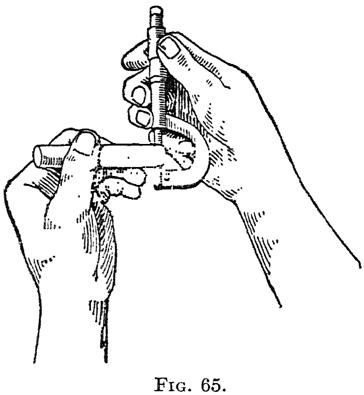


FIG. 65.

A thousandth of an inch is considered a very large amount in the estimation of a skilled mechanic. A good idea of the value of a "thousandth" may be had from measuring the large and small ends of a standard mandrel of ½ in. diameter or under. Measure the large end; measure the small end; subtract, and note that the difference is about 0.002 or 0.003 in. Again measure the large end and without changing the setting of the micrometer place it over the small end and note the "shake" between the measurement obtained at the large end and the diameter at the small end.

79. Holding a Micrometer.—Figure 65 indicates clearly the proper way to hold the micrometer in order to accurately

measure a piece held in the hand. Note carefully the position of the fingers; the micrometer is held by the little finger or the third finger, whichever is less awkward, against the palm of the hand, which allows the spindle to be operated in either direction with the thumb and index finger. The correct way to hold a micrometer when measuring work not held in the hand, is shown in Fig. 66.

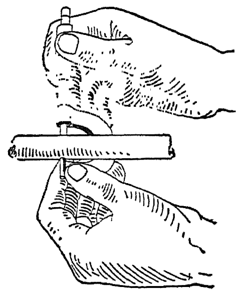


Fig. 66.

When making a measurement be sure the micrometer is held square across the diameter. Turn the spindle down to the work but not down too hard. It is easy to spring a micrometer 0.001 or 0.002 and this not only gives a false measurement but injures the micrometer.

It seems easy for some people occasionally to read a micrometer 0.025 over or under; such a mistake is inexcusable. It is even more careless to add 25 and 5 and call it 35, or 75 and 5 and read it 85. Be careful when using a micrometer to hold it properly, to adjust it carefully, and to read it accurately.

Questions on Measuring

1. How do you read a scale, using the significant graduations?

2. How do you hold a caliper to read the measurement on a scale?

3. What is the method of adjusting a firm-joint caliper?

4. What do you understand by sensitive touch?

5. In the limit gauge illustrated, Fig. 62*a*, what is the reason for rounding two corners and beveling the other two?

6. In the limit gauges illustrated in Fig. 62, why are the "not go on" gauging surfaces smaller than the "go on" surfaces?

7. What is the difference between the "go on" and the "not go on" dimensions of a limit gauge?

8. What part of an inch does one revolution of the thimble of a micrometer move the spindle? Why?

9. Into how many divisions is the hub graduated? How are these divisions numbered?

10. Into how many divisions is the beveled edge of the thimble graduated? How numbered? Why?

11. Describe the proper way to hold a micrometer (*a*) holding the work in the hand, (*b*) when the work is in the machine.

CHAPTER V

CENTERING

80. Holding Work in the Lathe.—Work to be turned in the lathe may be held between centers; fastened in a chuck; clamped to the large faceplate; or supported in the steady rest. Work that is to be faced or turned true with a finished hole is held either on a mandrel between centers; or on a

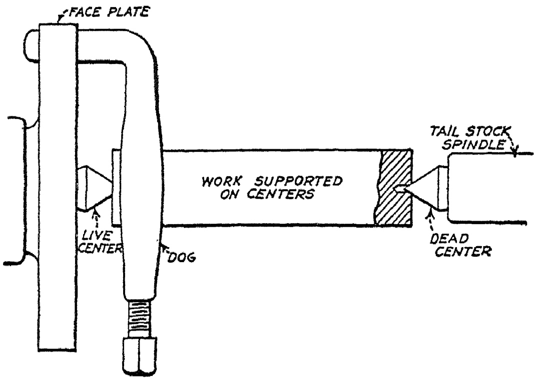


FIG. 67.

special arbor the shank of which fits the spindle; or on a plug which is fastened in the chuck and turned to fit the hole.

A large proportion of engine-lathe work is machined "on centers." Sixty-degree countersunk holes, called centers, are drilled and reamed in both ends of the piece to be turned. These holes fit the 60-deg. lathe centers and the work is thus supported (Fig. 67).

The work thus centered is usually driven from the small faceplate by means of a dog[1] which is securely clamped to it

[1] *Dog* (Fig. 68).—In lathe work, a driver. Made in many shapes and sizes and used to drive the work from the faceplate. The work is gripped by a setscrew and the finished end should be protected by a

on the live-center end. The work turns *with* the live center which acts as a support only, and *on* the dead center which acts as a support and also as a bearing (see Fig. 67).

81. Importance of Carefully Locating the Center.—It is very important to center carefully a piece to be turned. A piece carelessly centered may not have sufficient stock to clean, that is, to finish all over, and therefore be spoiled. Even if it has sufficient stock to clean, the fact that the centers are out of true will necessitate a big chip on one side of the diameter and

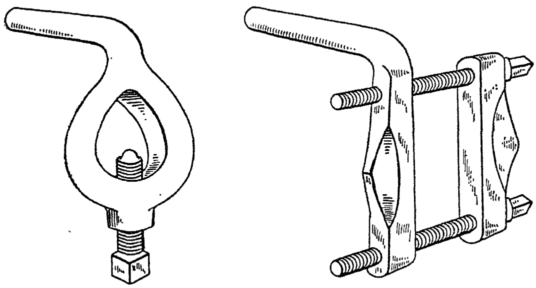


FIG. 68.—Lathe dogs in common use. *a*, bent-tail dog; *b*, clamp dog.

a small chip on the other. This unevenness of cut takes more time and may cause inaccuracy.

It is especially important to center tool steel carefully. In its manufacture the heating necessary for rolling the steel into bars of various shapes and sizes causes a decarbonization[1] of the outside to the depth of perhaps 1/32 in. If this is not altogether removed in machining a piece of tool steel, when the piece is heat treated, that part of the piece from which the decarbonized portion has not been removed will not harden.

The rolling process also causes a difference of density of the metal toward the center of the bar, and turning off more from

---

piece of copper or soft brass between the work and the hardened end of the screw.

[1] Tool steel has a definite content of carbon, usually about 1 per cent. When the steel is heated and exposed to the air, as in rolling, the carbon on the surface is lost and this surface layer is said to be decarbonized.

one side of the bar than the other will cause the piece to warp in hardening.

Care must be taken in centering cast iron. In making an iron casting the molten metal is poured into a sand mold. The

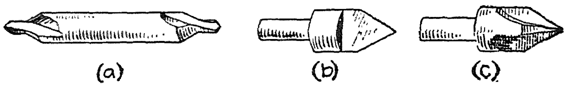


FIG. 69.

hot metal coming in contact with the cold sand causes the surface of the casting, the "scale" it is called, to become considerably harder than the interior. Also a considerable amount of sand is fused with the iron in the surface of the casting. These conditions serve to render the surface of a casting very hard and brittle, and when machining it is impor-

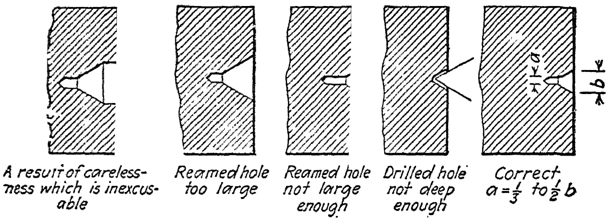


FIG. 70.

tant that the point of the cutting tool be well under this scale in order not to rub off the cutting edge and ruin the tool.

82. Size of Centers.—Combination drills and countersinks[1] are furnished in various sizes. There is no rule for the sizes of centers. The proportion of the size of the center to the diameter of the work is largely a matter of judgment and depends on the material, the amount of stock to be removed, the cut to be taken, the number and kinds of operations to be performed, and the shape of the piece.

[1] *Combination Drill and Countersink* (*a*, Fig. 69).—A tool which combines the center drill and the 60-deg. center reamer (or countersink). It is more efficient and has largely superseded the small drill, and the center reamers shown in *b* and *c*.

Do not make the centers too big; they should be just large enough to withstand the resistance of the cut. If, after facing, the center is too small it is easy to make it larger. The size of the *drilled* hole *a* (Fig. 70) should be about one-third to one-half the diameter of the *reamed* hole *b*. It should be sufficiently deep that the point of the lathe center does not touch thus ensuring a real bearing on the cone surface. This extra depth also provides a reservoir for oil.

For the preliminary cylindrical turning operations the following may be used as a guide:

Finished diameter of work, in.	Diameter of center, in.	Diameter of center drill, in.
1/2	1/8	1/16
1	3/16	3/32
1 1/2	1/4	3/32
2	5/16	

Too much emphasis cannot be placed on the importance of having correct centers—the right *shape*, the right *size*, *smooth* and *clean*.

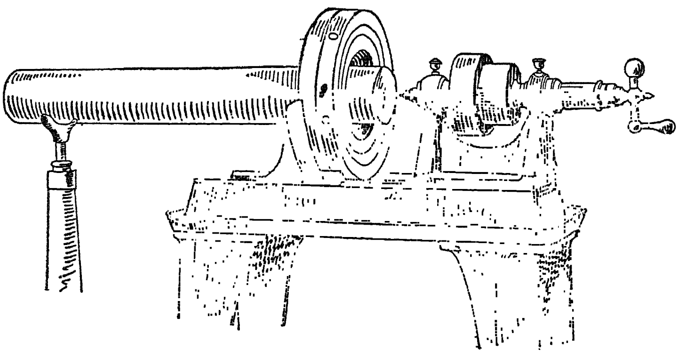


FIG. 71.—Centering machine. (*Courtesy of D. E. Whiton.*)

83. The Centering Machine (Fig. 71).—Modern machine-shop equipment includes a centering machine which automatically holds round pieces central with the drill spindle thus making it unnecessary to locate the centers otherwise. (There

are several kinds of these machines, some of which are provided with three jaw chucks and cannot be used for holding and centering square stock.)

In many shops a centering machine may not be available, and further, many of the larger and irregular-shaped pieces cannot be held in such a machine even if one is at hand, therefore other methods of locating the centers must be used. Some of these methods are here described.

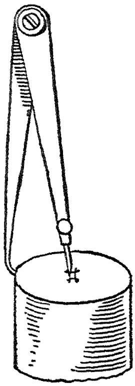


FIG. 72.

84. Methods of Locating Centers.—First, measure the stock to see if it will finish to the length required, and then rub chalk on the ends to make the center-locating lines show more distinctly.

Hermaphrodite-caliper Method (Fig. 72).—Set the caliper[1] to about the radius of the piece. Place the caliper leg on the circumference at the extreme end of the piece and with the point draw an arc near the center of that end. Move the caliper leg about one-quarter of the circumference of the end each time and draw three more arcs. The four arcs will form an approximate square, the center of which is the center required.

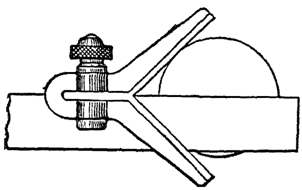


FIG. 73.—Center square. A blade so arranged with a square as to have one edge of the blade divide the angle of the square equally. It is used in scribing radial lines on the ends of a piece of work.

Center-square Method (Fig. 73). Hold both limbs of the center head of a combination set (Fig. 74) against the surface of the work and, with a scriber, rule lines at about right angles to each other. The intersection of these lines is the required center.

Surface-gauge Method (Fig. 75). When it is required to locate the center in an irregular-

[1] *Hermaphrodite Caliper.*—A combination of a caliper leg and a divider leg. Very useful for locating and testing centers and for laying off distances from an edge. Commonly called "morphy."

shaped casting or forging a surface gauge[1] may be used. For example: a bolt blank may be forged with the head offset. If the head is centered true the body of the bolt will run out of true and may not clean. To properly center such a piece

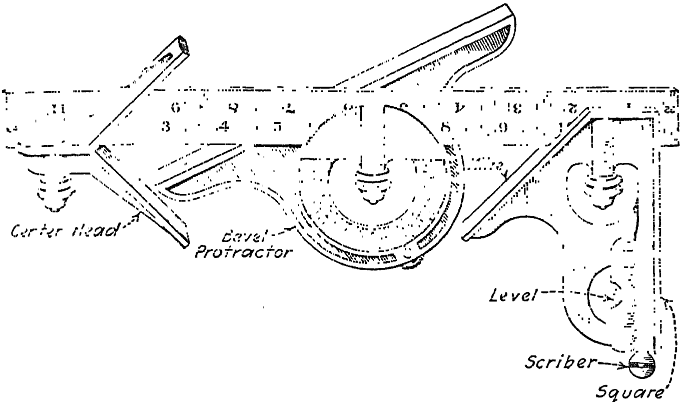


FIG. 74.—Combination set. Contains a rule, a square, a mitre, a center square, a protractor, a level, and a scriber. (*Courtesy of The L. S. Starrett Company.*)

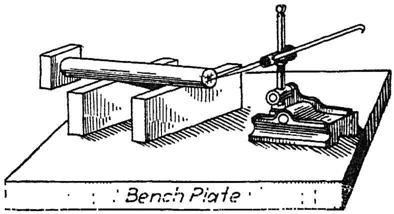


FIG. 75.—Surface-gauge method of locating centers.

place the bolt on parallels or on V blocks, adjust the surface-gauge scriber to the approximate center of the body, and draw lines on both ends forming squares. The centers of these squares are the desired centers.

Diagonal-line Method (Fig. 76).—Rectangular pieces are easily centered by drawing diagonal lines.

[1] *Surface Gauge.*—A tool with a heavy base supporting a sharpened scriber which is adjustable as to height. It is used for gauging, marking, or transferring the distance from a flat surface to a point or line on the work.

85. Use of Center Punch.—After the center is located, catch the piece in a vise and select a center punch (Fig. 77) with the point ground to about 90 deg. Tap lightly at first, being sure that the point is central, then make the indentation large enough to support the piece while it is being tested between the lathe centers. It is important to test the accuracy of the center punch marks, especially until experience has trained the eye. Do not put on a dog, or do not start the lathe, but revolve the work by hand on "dead centers," marking the "high spot" on each end with chalk.

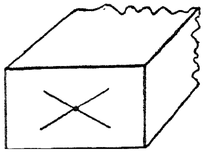


FIG. 76.

1. Do not tighten the dead center too much, but allow the work to spin freely.

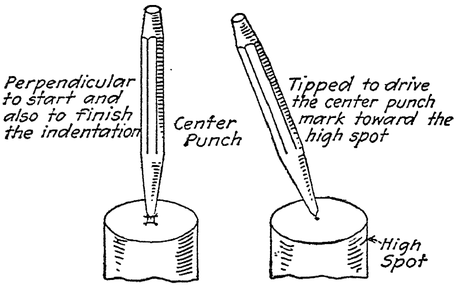


FIG. 77.

2. Spin the work slowly at first to see if it runs out of true; if you spin it too fast the blur makes it appear to run true.

3. If it does run out too much, then spin it fast and with a piece of chalk just touch the spinning work about half an inch from each end. The chalk will not mark all around the piece but will mark the "high spot."

Catch the work in the vise again, and by tipping the center punch, drive the center toward the high spot to bring it right. Then hold the punch perpendicular and make the indentation symmetrical. Where the chalk marks the piece

(the "high spot") indicates the greatest radius. To shorten this radius serves to bring the center punch mark towards the center. Sometimes the beginner may need to change the center punch mark and test the location two or three times before the work runs true enough.

86. Aligning the Lathe Centers.—The dead center and live center should be in line for all operations except taper turning. Form the habit, when going to work on a lathe, of making sure that the centers are in approximate alignment.

To ascertain if a lathe is in approximate alignment: (1) Note that live center runs true and carefully bring the dead center to within $\frac{1}{16}$ in. of the live center. A very little offset will be easily seen. (2) "Witness marks" on slide and base of tailstock will usually indicate whether or not alignment is close. Setting a lathe in *accurate* alignment is explained on page 137.

87. Drilling and Reaming the Center.—The combination drill and countersink (*a*, Fig. 69) is the correct tool to use for making the center hole.

1. Obtain a combination drill and countersink of the right size and be sure it is sharp. Get also, a small drill chuck (Fig. 114, page 160) with a taper shank that will fit the taper hole in the main spindle of an engine lathe (or speed lathe).

2. See that the centers are in approximate alignment.

3. With a suitable steel rod through the spindle hole knock out the live center, holding it so it will not fall.

4. Be sure the taper shank of the chuck is clean and fits the taper hole, then thrust it tight in place.

5. Fasten the combination drill and countersink firmly in the chuck.

6. Let the tail spindle project about an inch, move the tailstock to such a position that the work may be held between the point of the drill and the dead center, and then clamp the tailstock to the ways.

7. Put one of the center punch marks on the drill point and, steadying the work, bring the dead center carefully into the other center punch mark.

8. Hold the work fairly tight with the left hand, allowing the tool rest to support the wrist.

9. Apply lard oil or cutting compound for steel. Drill cast iron dry.

10. Start the lathe (fastest speed), and feed by turning the tail-spindle handwheel slowly until the center hole is reamed to the correct size.

11. Be careful not to break the drill when drawing the work back from the center reamer. It is best to keep the work against the tail center as it is backed away from the drill, thus avoiding any tendency to pry off the drill.

88. Removing a Broken Center Drill.—If the drill should be broken in the center hole it may often be removed by a sharp blow on the side or end of the piece of work; if not it must be softened by annealing[1] when it may be drilled out.

Questions on Centering

1. Does it make any difference whether the "morphy" is set a little more or a little less than the radius?

2. Why do you rub chalk on the ends of the stock?

3. How large an indentation do you make with the center punch before testing? Why? What is the angle of the point of the center punch?

4. After both centers are located and center punched, how are they tested for accuracy? What does the "high spot" indicate?

5. If the center punch marks need changing slightly, how is this done?

[1] *Anneal.*—To reduce brittleness and increase softness (also eliminates strains in steel caused by forging and rolling).

To anneal carbon tool steel (*usual method*): Heat thoroughly to a dull red; then cover with ashes, powdered charcoal, lime, or asbestos, and allow to cool (see also p. 365).

"Cold water" annealing.—Heat to a dull red; then allow to cool slowly until no red can be seen in a fairly dark place; then plunge into cold water. It is much quicker than the usual method, but do not attempt cold water annealing for high-speed steel or for carbon steel over ½ in. in thickness.

To anneal high-speed steel: Heat *very slowly* to a "bright cherry color" (1450 to 1500°F.), and cool *very slowly* in the furnace itself, or in ashes, lime, or asbestos. Annealed high-speed steel is somewhat harder than annealed carbon tool steel.

6. Why should care be taken to have the centers in the work fairly true?

7. Why is the combination drill and countersink an efficient tool?

8. How many r.p.m. should the combination drill and countersink be run?

9. Name at least two things that determine the proper size of the centers to be reamed.

10. Why does the drill break so easily? What is a good way to pry it off?

11. How tightly do you grasp the work when you are drilling it? Where do you rest your hand? Give reasons.

12. On what materials is a cutting lubricant used when drilling or reaming? What is the use of the lubricant?

13. How is the center square used for finding the center?

14. If the nature of the work is such that the use of a "morphy" of a center square is not practicable, for example, a bolt head that is forged off center from the body, how may the centers be located so that the body will run true?

15. What is a centering machine? What are its advantages?

16. Give two reasons for the drilled hole being deeper than the point of the lathe center.

17. If a drill has been broken off in the center hole, why must it be annealed before it can be drilled out?

CHAPTER VI

FACING

89. Facing: General Information.—The lathe operation of finishing the ends of the work, to make the ends flat and smooth, and to make the piece the required length is called facing, in some shops "squaring."

The work may be held on centers or in a chuck.[1] An advantage of holding the work in a chuck lies in the fact that the work does not have to be centered before facing, consequently one end may usually be faced clean and then all the overlength taken off the other end without worrying about the relative size of the centers. Also, a shoulder tool or a regular turning tool may be used, and this is often an advantage especially when removing the scale from cast iron. A disadvantage is the time it takes to put on the chuck and afterwards to remove it. Further, many pieces cannot be held in a chuck and faced because they would project too far; 3 or 4 in. is the limit.

The term radial facing is applied to work of a comparatively large diameter and may more properly be discussed under the subject of chuck work (see page 165).

The ends of a properly faced piece will be square (flat). The surface may be tested with the edge of a rule. To produce a flat surface when facing on centers the lathe centers must be in line; a surface like *a*, Fig. 78, indicates that the dead center is offset towards the operator, a surface like *b* shows the center is offset away from the operator.

If considerable material is to be removed later in the turning operation it is better not to face to exact length until all the rough turning cuts are taken. This will provide a larger

[1] For information concerning chucks, see page 160.

center for the roughing cuts than would look well in the finished piece. Also, finishing the ends after roughing out the work will remove the burr around the center caused by the great pressure of the heavy roughing cut that tends to enlarge the hole as the work revolves.

A piece of steel is usually cut off somewhat over the finished length because the hack saw or cutting-off machine of any kind is not meant for accuracy. The machinist before starting to work on any special piece measures the stock to be sure it is long enough. If there is any doubt about the stock being long enough, a "telltale" (an unfinished portion of the end) is left *on each end* of the piece to prove, if necessary, that it was short.

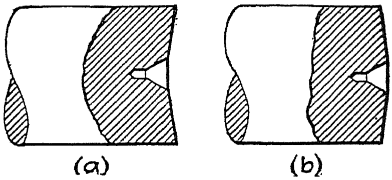


Fig. 78.

Suppose the work is to be faced on centers. Face off about half the overlength from one end; remove the dog; lay off the length (with the hermaphrodite caliper if the piece is short, otherwise with rule and scriber); place the dog on the end that has been faced and finish the other end practically to the line. If a chuck is used instead of centers, just as much care must be used in laying off and measuring.

An accurate way of measuring the length of the shorter pieces is with an outside caliper. Sometimes on the longer pieces it is necessary to use a rule to measure direct or possibly two rules end to end. When using this method one must be particularly careful to have the end of the rule flush (even) with the end of the work. A hooked rule (Fig. 79) is very useful.

Remember always that the function of the tool is to cut the metal, not scrape it. Do not take a great number of chips. Learn as quickly as possible to judge the facing chip that will remove a given amount of stock.

Facing accurately and quickly is a job which requires attention to business and good judgment. The beginner who with reasonable speed can face a piece on centers to length and have the ends square and smooth, with no ridges, no undercut

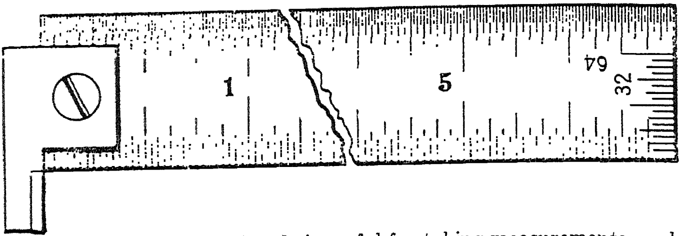


FIG. 79.—Hooked rule. This rule is useful for taking measurements, and for setting or reading the measure of an inside caliper. (*L. S. Starrett Company.*)

near the center, and no fin, has a right to be proud of his work.

90. Facing on Centers: Adjusting the Work.—After the work is carefully centered, tighten a dog firmly on one end (a protecting piece of copper under the dog screw is unnecessary unless that end of the work is finished). Put a drop of oil in the center hole in the other end. Place the work between centers and *adjust* the tail center. The experienced machinist always puts the work on the live center first, being sure the tail of the dog does not bind in the slot of the faceplate (Fig. 80). Then he runs the tail center into the center hole, and is careful that it enters without hitting around the hole two or three times. To avoid chatter, he brings the center up until there is no shake of the work between centers, but *not too tight* because the work must be free to turn or the center will be scored and possibly spoiled. That is, he *adjusts* the work between the centers

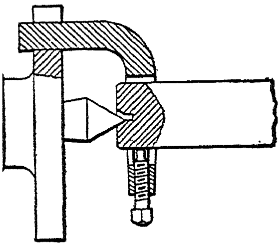


FIG. 80.—When using a dog larger than necessary it is liable to bind on the sides of the faceplate slot, and may also bind on the bottom. In either case the work will not fit on the lathe center and will not run true.

carefully. Unless the work is too large, the machinist always tries this adjustment by wiggling the tail of the dog in the faceplate slot.

When placing the work between centers do not hold it at arm's length or try and reach over the tool post. Hold the piece steady by resting the hand against the tool rest or the tail spindle or in any convenient way.

91. The Facing Operation.—A side tool is used for facing. There are several patented holders with bits or blades for facing, but perhaps the regular turning-tool holder with a

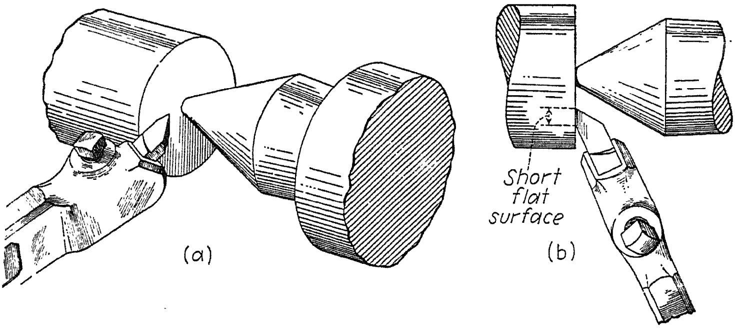


FIG. 81.—Facing or "squaring."

bit ground as shown in Fig. 51, page 89, is as satisfactory as any of these.

The whole cutting edge of the side tool should not be set at right angles to the center line of the work, but the point of the tool should be slightly in, that is, toward the work.

To obtain a smooth finish the point of the side tool should be slightly rounded with an oilstone, or so ground as to present a short flat surface to the work (see Fig. 81). The length of this flat surface should be greater than the amount of the cross feed.

To make sure that the flat spot is set square, take a light facing cut with the tool set fairly near right "by eye." Then run the tool in toward the center and also a trifle away from the end of the work. Now, watching the cut, carefully feed the tool (hand long feed) to just touch the work and note

if the "heel" and "toe" of the flat spot both touch on the squared surface. If they do, the flat spot is set square, if they do not, it will be easy to reset the tool.

The work may be roughed by feeding from the center hole outward or from the circumference toward the center. *The finishing cut is made from the center hole outward.* (If considerable overlength is to be removed by facing, it may be advisable to rough by feeding the tool *sideways*, that is, "stepping off" the excess material by several hand-fed side cuts instead of radial cuts, and perhaps a turning-tool bit properly ground for this purpose may be more efficient.)

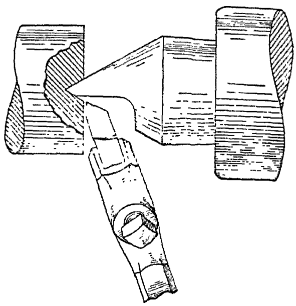


FIG. 82.—The use of a half center for facing.

If there are several pieces to face, it will be advisable to get a half center (Fig. 82) from the toolroom, but having only one or two pieces, or in the event of a half center not being available, the following method is used.

Adjust the work between the centers (no slack and not too tight), start the lathe, and with the right hand on the cross-feed handle and the left hand on the long-feed handwheel, bring the tool as close to the center as convenient, and then into the work until a light chip is taken around the center hole. Stop feeding with the left hand (long feed) and with the right hand feed slowly outward, that is, toward the circumference, thus squaring or facing the end. The amount of chip and feed must be determined by conditions. The machinist usually prefers to face the smaller pieces (under 1 in. diameter) by hand feed. The beginner will go slowly on the first piece.

The above method of facing will leave a slight fin[1] around the center hole. To remove this fin bring the side tool to the

[1] Sometimes the machinist purposely leaves a fairly heavy "fin," say 1/4 in. or more in diameter, around the center hole until he is ready to take the finishing cut. He judges by the looks (length) of the fin how much he has removed from the end of the stock.

surface just faced and to the fin. Ease the tail center about $\frac{1}{32}$ in. and clamp the tail spindle; when the lathe is started the work will have a tendency to ride on the tool and make a clean cut around the center hole. Do not feed toward the circumference again but stop the lathe and remove the work.

92. A Typical Facing Job.—Suppose there are a dozen pieces to be faced and that they may be held either on centers and use a half center, or in a chuck. If the work may be held conveniently either way, there is not much choice between the methods except that if the centers are used, care must be taken not to face too much off one end and thus leave the center holes unequal in size. In either case, on centers or in a chuck, a facing tool or a shoulder tool (*C*, Fig. 51) may be used. Proceed as follows:

1. Lay the pieces side by side, ends flush (even); pick out the shortest one and measure the length in order to judge how much to face off one end of this and succeeding pieces.
2. Set the tool with the facing *flat spot* square.
3. Face one end of each piece. If a shoulder tool is used, the roughing cut may be made by feeding from the circumference toward the center. Then face "in" a trifle with the same tool and feed from the center out to give a smooth finishing cut.
4. When all of the pieces are faced on one end, lay off the lengths and face the other ends; the roughing cuts very close to the line, the finishing cuts splitting the line.

Questions on Facing

1. Why should the piece be measured before proceeding to face?

2. It may be stated that a side tool has four clearance angles; where are they? What three are ground on the tool?

3. If a patent holder is used, why should the blade be removed from the holder before grinding?

4. If it is necessary to sharpen a side tool, should it be ground on the top, on the front, or on the side? Why?

5. Should a side tool be oilstoned? Give reason.

6. Why must the centers be in line? What is the effect if the tailstock slide is offset toward the operator? Away from the operator?

7. How do you adjust the work between centers?

8. Do you always finish one end before rough facing the other end? Give reasons.

9. What is a "telltale"?

10. If you have several pieces to face, is it advisable to rough face them all before finishing any? Give reason.

11. When is hand feed used in facing? When is power feed used? Give reasons for both.

12. Is the tool fed in a direction toward or away from the center? Why?

13. Why is it advisable to slightly round the point of the side tool?

14. What do you mean by setting the side tool "on center" and pointing "slightly in" toward the work?

15. For a facing cut, how do you lay off the length? How do you measure the length?

16. When should a piece not be faced to exact length before the turning operation?

17. When facing an end, one roughing cut and one finishing cut are usually sufficient. Why take more?

18. If considerable metal is to be removed what tool other than a side tool may be used? Why is it more efficient? How may it be fed?

19. What causes a wavy or chattered appearance of the surface being squared?

20. What is a half center? Why is it more than half?

21. What is a "fin"? What is the best way of removing a fin? Another way?

22. When is a lubricant used in facing?

23. Name at least two reasons why a piece is faced.

CHAPTER VII

TURNING IN A LATHE

93. Introduction.—Turning in a lathe is accomplished by causing the work to revolve while the tool being fed longitudinally peels off a chip. The same definition may be applied to turning in a turret lathe or to turning outside or inside (boring) in a "boring mill." The principles and methods involved in turning and boring are the same for the larger machines as for the smaller machines. The cutting action of the tool is the same whether the work is held between centers or in a chuck; on the faceplate of a lathe; on the table of a boring mill; or in a special fixture in either machine. The principles of feeds and speeds, of alignment, of measurement, etc., are fundamental machine-shop principles.

In rapid-production work where hundreds or thousands of duplicate pieces are made within narrow limits of exactitude, the turret lathe with its special holding fixtures, its facing, turning, boring, and other tools, each arranged in the turret head or on the cross slide for its particular operation, is most advantageously used. For the larger castings and forgings the boring mill is most adaptable for many turning, facing, or boring operations. It must be remembered that these machines are after all modifications of the lathe. The lathe is the most important machine; it is the most widely used and the most adaptable of machine-shop tools and turning in a lathe is one of the most important operations in a machine shop.

In order to be able intelligently to set up a lathe for an accurate turning job it is necessary to understand certain of the *principles* and *methods* involved.

PRINCIPLES OF TURNING

94. Position of the Dead Center.—The center line of the lathe is determined by the center of rotation, or the axis, of the

main spindle, and is parallel to the ways. It is therefore parallel to the line of travel of the carriage which moves on the ways, and also parallel to the line of travel of the turning tool.

The live center and the dead center are equidistant from the horizontal plane of the ways. The live center, having no adjustment, is fixed in its position.

The design of the tailstock, however, permits of transverse adjustment of the dead center, that is, the dead center may be

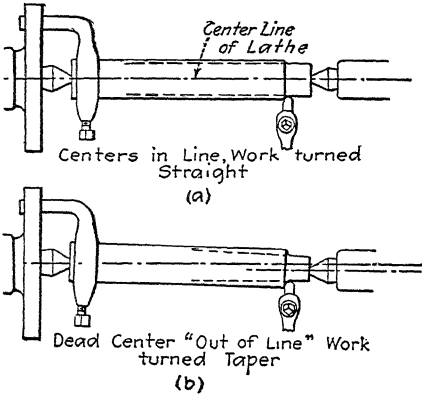


FIG. 83.

moved *off center* toward or away from the operator, or it may be adjusted from off center to an exact central position.

When turning work in a lathe, if the dead center is exactly *in line* with the live center, the distance from both centers to the line of travel of the tool is the same. In this case the radii and consequently the diameters of the turned work are everywhere equal, and the turned piece is a perfect cylinder. It is said to be *turned straight* (see *a*, Fig. 83).

If, when turning, the dead center is *offset* or *out of line*, the distance from the line of travel of the turning tool to one center is greater than it is to the other, and the diameter of the work which is being turned changes constantly from one end to the other and the work is not straight. It is said to be

turned taper (see *b*, Fig. 83). Methods of obtaining accurate alignment of centers are explained beginning page 137.

95. Accuracy of the Live Center.—The method of procedure, in turning a cylinder in a lathe, is to turn half its length or more from one end, then reverse the piece on centers (changing the dog to the opposite end of the work) and turn the rest of the cylinder. The circumference of any properly turned piece is concentric with its center line. If the live center runs true and a piece is turned as above, it will be found that the part of the cylinder first turned will run absolutely true when reversed, and when the remainder is turned the two cuts will meet exactly. This work is right, and is possible only when the live center runs perfectly true.

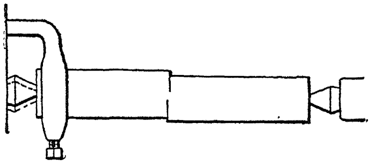


FIG. 84.—Result of attempting to turn a cylinder with the live center "out of true."

Suppose that the live center runs out and the above operations are made. When the piece is reversed it will be noticed that the middle of the work runs out one-half the amount the live center is out of true. It is impossible to have the cut which is made after the piece is reversed meet flush with the cut already made and the piece cannot be turned straight to size (see Fig. 84). Further, suppose the live center runs out of true and a portion only of the length of the work is turned. If the center does not run out too much, and the stock is large enough to clean, the turned portion will be round and straight *but it will not run true* on dead centers and it will not run true in a lathe or other machine in which the centers are true. Many times it has been discovered after several later operations (in which the centers were not used) that the work has been spoiled in turning; this means delay and waste. The live center *must* run true.

How to Determine if the Live Center Runs True.—One method of determining if the live center runs true is as follows: Tighten a toolholder (reversed) in the tool post and

bring the end fairly close to the center; hold a piece of paper under the revolving center and look down between the tool-holder and the center. Any eccentricity of the center may readily be observed.

96. Cleaning and Truing the Lathe Centers.—The shanks of the lathe centers are taper and fit taper holes in their respec-

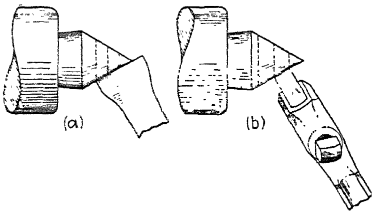


FIG. 85.—Truing a line center with (*a*) forged square-nose tool and (*b*) square-nose tool bit.

tive spindles. Any dirt, chips, or burrs, either on the shank or in the taper hole, will cause the center to run out of true. Clean the center with the palm of the hand, and if any nick or burr is felt, oilstone it off. Clean the tail-spindle hole with

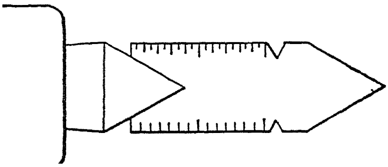
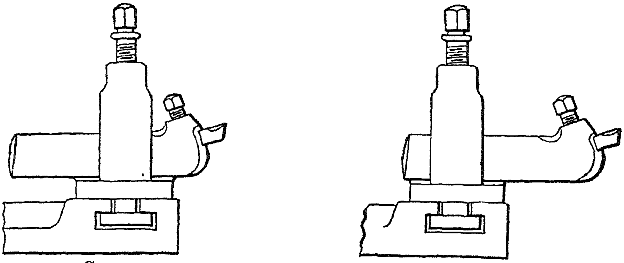


FIG. 86.—Center gauge. A tool used for testing the accuracy of 60° centers. Used also when grinding and setting 60° thread tools.

clean waste on a stick, or with your finger. To clean the main-spindle hole, push a piece of waste through the whole length and wipe the taper hole with your finger *but never while the lathe is running. Never clean any moving part of any machine.* If the live center runs out after cleaning, it may be trued up with a square-nose tool (Fig. 85) or with a turning tool if the compound rest is used (see page 212). A center should be filed very little, if at all, after turning. Test the angle of the center with a center gauge as shown in Fig. 86.

Some lathes have been abused to such an extent that the taper hole in the main spindle runs out of true. In such a case it is necessary to have a "witness mark" (perhaps a center punch mark) on both the spindle and center, and bring these two in line when putting in the live center.

The dead center is hardened and if damaged, must be ground.[1] It should be smooth. The dead center is a bearing



Correct
Tool caught short with only sufficient room to use wrench for removing tool point.

Incorrect
Too much leverage permits spring and vibration, causes chatter and inaccurate work.

Fig. 87.

on which the work revolves, therefore *it must be kept well oiled* or it will run dry in the center of the work and become roughened and probably twisted off.

Some machinists advocate a hardened live center. It is no doubt stronger than a soft center. To have the live center, either soft or hard, run *true*, it is usually best to true it when it is in position in the lathe spindle, and therefore in the case of a hard center, it is necessary to use the special grinding attachment. For most operations the soft live center is satisfactory.

[1] *Grinding Centers.*—There are several styles of grinding attachments that are designed especially for truing centers. If one of these attachments is not available the dead center may be cold-water annealed and trued up, and then rehardened (for hardening see page 340). Such a center is good enough for the dead center but is not likely to be true enough, after hardening, for the live center (for annealing see page 347, also footnote page 118).

97. Setting the Tool.[1]—1. *Catch the tool short* (see Fig. 87). The farther the cutting edge of any tool projects from the tool post the greater the leverage and the more the spring of the tool. This causes chattering and often worse evils and should be avoided wherever possible.

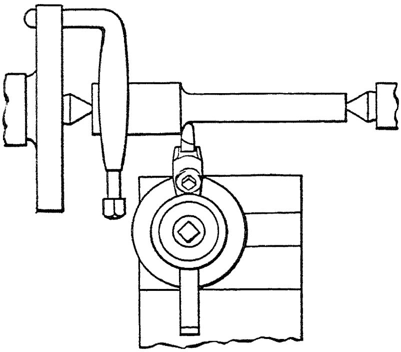


Fig. 88.—Shows tool post on left-hand side of tool rest. In this position there is less tendency to feed so far that the dog will strike the tool rest.

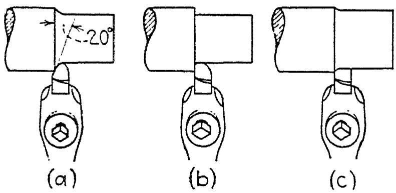


Fig. 89.—The cutting edge as presented to the work. (*a*) Most efficient turning tool; (*b*) does not remain sharp as long or produce as good work as (*a*); (*c*) very inefficient except for certain finishing cuts.

2. The tool post should be located at the *left-hand end* of the T-slot in the tool rest (Fig. 88). If it is clamped in the middle or right-hand end of the slot the danger of the dog hitting the tool rest is greatly increased. If the dog hits the tool rest when the lathe is running, the point of the live center will be broken off and the center hole in the work spoiled.

[1] It is assumed that the beginner in lathe work will be provided with properly sharpened cutting tools until such a time as it is convenient to have him learn to grind them for himself. Information concerning cutting tools for lathe work is given in the text, beginning page 74.

3. The position of the cutting edge as presented to the work has a considerable influence on the finished appearance of the work and also on the life of the tool. It may be easily proved that a cutting edge at right angles to the center line of the work will not cut so efficiently as when arranged at an angle of about 20 deg. with this perpendicular (see Fig. 89).

4. The usual practice with modern lathe turning tools (toolholders with high-speed tool bits) is to set the tool point toward the dog (see *a*, Fig. 90) thus obtaining approxi-

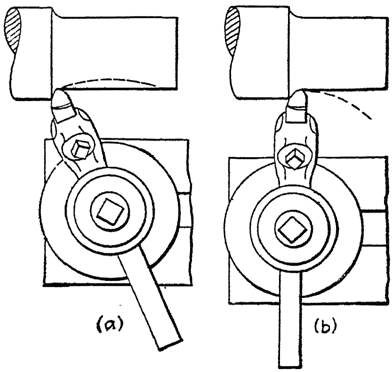


FIG. 90.

mately the proper side rake with less grinding on the top of the tool bit. This is all right for light cuts if care is taken to tighten securely the tool in the tool post. For heavy cuts, however, it is best, if possible, to have the tool point a little away from the dog for if it slips it will move away from the work and not into it (see *b*, Fig. 90).

5. Have the point of the tool on center or a trifle above in order to have the correct amount of front clearance (this is illustrated in Figs. 49 and 50, page 86).

98. Direction of the Feed.—In machining a piece of work on centers the feed should be toward the headstock because then the pressure is on the live center, which revolves with the work. The dead center is a bearing on which the work

revolves and if a heavy feed is directed against it, undue friction will result which will tend to score the center hole in the work and may possibly twist off the end of the lathe center.

99. The Use of a Protecting Piece.—A finished surface that has been scored or dented by the sides of the dog or by the dog screw is an indication of extreme carelessness or ignorance. To avoid that sort of thing, the machinist uses a protecting piece of copper or soft brass of reasonable thickness. He does not use sheet steel because steel is too hard and the dent the dog makes in it is carried through into the work. He does not use paper or cardboard because these are too soft and the dog screw crushes through.

100. Adjusting Work on Centers.—It is quite necessary in the beginning of one's machine-shop experience to acquire the habit of making a proper *machine setup* and *work adjustment.* It is one of the habits that distinguishes an expert on any machine in any shop.

Have the tail spindle projecting about 3 in. (usually) and move the tailstock to about the distance from the headstock necessary to hold the work between the centers and then clamp the tailstock to the bed.

Tighten the dog securely, using a protecting piece of copper or brass if necessary, and put a drop of oil in the center hole. Put the other center hole on the live center, and holding the work against the live center with the left hand supported on the tool rest, run the dead center into its center hole. Then adjust the work carefully, wiggling the tail of the dog to make sure the adjustment is tight enough but *not too tight.*

101. Oiling and Readjusting the Centers.—It is very important to keep sufficient oil (or white lead) on the dead center. The heat generated in turning, especially during a heavy roughing cut, will quickly burn up the oil on the bearing surface and a dry bearing always cuts. Also the heat of a heavy cut may expand the work enough to unduly tighten it between the centers. It is not enough to squirt some oil at the center: stop the lathe, draw back the center, put the oil where it will do the most good, and *readjust* the dead center. A dry bearing

will sometimes give warning by a faint squeak. *Always investigate a squeak.*

A *live* tailstock center with antifriction bearings has been developed for high-production lathes (see Fig. 40, page 71). It must be remembered, however, that even though this center turns with the work and oil in the center hole is not required, the heated work expands just the same, and if the expansion is such that the work lengthens appreciably, this, as well as any other tailstock center, must be kept in mind and readjusted if necessary.

102. Lubricating the Tool.—There is no doubt that a good flow of oil or cutting compound on a turning tool will make

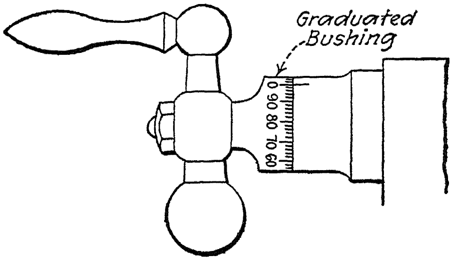


Fig. 91.—In most machines the graduated bushing may be loosened and then set at zero or elsewhere as desired.

for longer life of the tool, more work, and a better finish, when machining steel or wrought iron. Most engine lathes, however, are not equipped to obtain a flow of oil and the small amount that can be applied with a brush is not usually considered worth while except when cutting threads.

103. Graduations on Cross-feed Screw.—Most lathes are equipped with a graduated bushing on the cross-feed screw which will show in thousandths of an inch the movement of the cross slide. For example, the cross-feed screw in Fig. 91 has 10 threads per inch; one complete turn of the handle advances the cross slide $1/10$ in. The bushing is graduated into 100 equal divisions, therefore a movement of one of these divisions past the line on the nut (shown in line with the 0 graduation on the bushing) will indicate a movement of the cross slide of $1/100$ of

$\frac{1}{10}$ in. or $\frac{1}{1000}$ in. Remember that this movement affects the *radius* of the work and is *doubled* on the diameter.

Caution.—All lathes are not *graduated* to read thousandths, but the *numbers* stamped on the graduations read thousandths. For example: if there are 10 graduated spaces between the numbers 0 and 10 the graduations read thousandths but if there are only 5 graduations between 0 and 10 then each graduation will indicate 0.002 in. When going to work on a strange machine note the graduations, whether they indicate 2 thousandths, 1 thousandth, or $\frac{1}{2}$ thousandth.

104. Lost Motion in the Cross Feed.—There is always "back lash" or lost motion between any freely revolving screw and the nut. The amount of lost motion depends on the looseness of the thread in the nut and is of course increased by wear. The amount of lost motion in the cross-feed screw in a new lathe may be 0.005 in. and in an old lathe 0.020 in. or more. Suppose the lathe operator runs the cross slide in 0.005 in. too far and then merely moves the handle back 0.005 in.; the cross slide has not moved and will not move until the lost motion is taken up. The best way to correct an error of this kind is to move the handle back more than is really necessary, then take up the lost motion in the other direction and feed in again to the proper mark.

105. Cutting Speed and Feed.—One of the most important principles in lathe work, or any machine work, is involved in cutting the metal at the proper speed and feed. The reader is referred to page 95.

AN EXAMPLE OF TURNING

As an example let it be supposed that a machine steel cylinder is to be finished $6\frac{1}{2}$ in. long and $1\frac{3}{8}$ in. in diameter. The stock furnished is $1\frac{1}{2}$ in. in diameter and $6\frac{9}{16}$ in. long. Be sure the live center runs true and note that the lathe centers are approximately in line. Center the work carefully (centers about $\frac{3}{16}$ in. in diameter) and, in this case, face to length. The next operation is rough turning. In order to turn *straight* it is necessary to have the lathe centers exactly in line.

106. Alignment of Centers.—With the turning tool take a cut (No. 1, Fig. 92) quite near the dog just deep enough to get under the scale and about 1/4 in. wide. *Without changing the position of the cross feed,* move the dead center away from the work, swing the piece clear from the turning tool, and run[1] the carriage back to take cut No. 2. Put the work back on the centers and take cut No. 2. Caliper both cuts. If they are of the same diameter the lathe is turning straight because the two cuts indicate exactly the same effect as if one cut had been made the whole length to the dog. If the two cuts do not measure alike and cut No. 2 is the larger diameter, the tailstock slide should be moved toward the operator. If cut No. 2 is

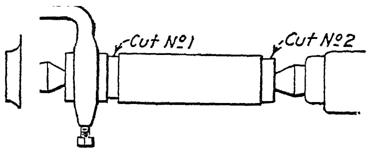


Fig. 92.

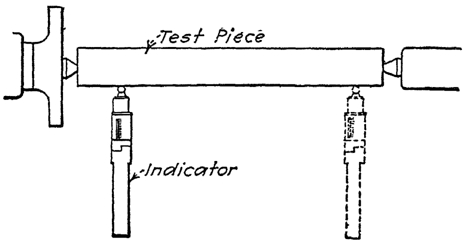


Fig. 93.—Best method of aligning centers.

the smaller the tailstock should be moved away from the operator.

107. Quicker Methods of Aligning Centers.—If a "test piece" (a piece 8 or 10 in. long, turned round and straight) and an indicator[2] are available the lathe centers may be quickly

[1] "Run" means to cause to perform a characteristic motion, hence the shop terms "run the tail center back"; "run the carriage back"; "run the tool in," etc.

[2] *Indicator.*—An instrument which with multiplying levers plainly shows the slightest movement of the point (see Fig. 94). There are many kinds of indicators ranging in price from a few cents to several dollars. The principle involved in all of them is, that the amount the

aligned. Place the test piece on centers without a dog and the indicator in the tool post. Run the indicator along the length of the test piece (Fig. 93) and when no movement of the indicator needle is observed the centers are in line. If no indicator is available, lightly pinch a piece of paper between the butt end of a toolholder and the test piece near one end and then near the other end and note the graduations on the cross-feed screw for each position. The reading of the graduations will indicate whether or not the centers are in line.

108. Setting the Speed.—The next step is to get the proper number of revolutions of the spindle to give a correct cutting speed (in this case, with a high-speed tool, say about 70 ft. per min.). The formula is:

$$\text{R.p.m.} = \frac{\text{cutting speed}}{\text{dia. of work} \times 0.25}$$

Substituting values and solving:

$$\text{R.p.m.} = \frac{70}{1.5 \times 0.25} = 186$$

Then fix the spindle speed as near 186 r.p.m. as possible.

109. The Roughing Cut.—Tighten the dog securely on one end of the work, put a drop of oil in the opposite center, and adjust between the lathe centers (no slack and not too tight).

With the tool post at the left-hand side of the tool rest set the tool well back in the tool post. For roughing, it usually

---

work is out of true is magnified many times and indicated by a needle, usually on a dial reading thousandths of an inch. An indicator is intended for use in setting centrally any point or hole in a piece of work to be operated upon in a lathe or upon a faceplate, also for testing lathe centers, shafting, and other work held between centers, the inside and outside diameters of cylinders, pulleys, etc., and work of a similar nature.

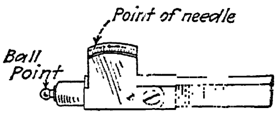


Fig. 94.

It is also very useful when aligning vise jaws or angle irons in shaper, planer- or milling-machine work or in aligning a finished surface preparatory to taking cuts which are to be parallel or at right angles to this surface. Other indicators are illustrated on pages 272 and 273.

is set square or pointing a little to the right. In this case, work $1\frac{1}{2}$ in. in diameter, set the tool about $\frac{1}{32}$ in. above center.[1]

An accurate job is required so it will be necessary to take two cuts, a roughing cut and a finishing cut. The diameter is to be reduced $\frac{1}{8}$ in. (stock $1\frac{1}{2}$ in. to finish $1\frac{3}{8}$ in.), that is, the depth of cut is less than $\frac{1}{16}$ in. so a coarse *feed* (say up to $\frac{1}{32}$ in.) may be used without strain and without tearing the surface.

Leave about $\frac{1}{32}$ in. for the finishing cut. Being careful not to turn undersize, feed by hand for $\frac{1}{8}$ or $\frac{3}{16}$ in., stop the lathe, and measure the cut. When the diameter measures about $\frac{1}{32}$ in. oversize, throw in the feed and turn about half the length. Throw out the feed first, then stop the lathe, but do not touch the cross-feed handle. Now take out the work and without moving the cross feed run the carriage back to the beginning of the cut. Change the dog end for end on the work, and now use a protecting piece under the screw. Adjust the work on centers again and turn to meet the cut already made.

110. The Finishing Cut.—Approximately $\frac{1}{32}$ in. is left on the diameter for finishing. The amount to leave for the finishing cut depends largely on the character of the roughing cut (a very coarse roughing feed has a tendency to tear the surface even if a fairly sharp tool is used). At least $\frac{1}{64}$ in. should be left for the finishing cut ($\frac{1}{32}$ in. on the diameter) because less than $\frac{1}{64}$ in. does not give the cutting point a chance to get under the chip and the result is a rubbing or burnishing effect which rapidly dulls the edge and produces a poor finish. The feed for finishing is finer than for roughing, say $\frac{1}{64}$ in. or less for this job.

A keen cutting edge is essential for a good finish. If several pieces have been roughed with the tool it may be necessary to

[1] Until the beginner's eye has been trained, it may be better to set the tool before putting the work in the lathe, and to use the point of the center to gauge the height of the tool. The trained machinist sets the tool after adjusting the work.

regrind it but if only one or two probably a few rubs with the oilstone is sufficient. Adjust the cross feed until the tool just touches the rough-turned surface of the work, then run the tool off the work toward the dead center. Using the graduations on the cross feed run the tool in, but not quite the full amount. Take a cut wide enough to caliper and measure the work again. (NOTE.—This precaution takes only a moment and is always advisable.) Having noted the extra amount to be removed start the lathe, throw out the feed, run the tool off the work again, move the cross slide in the required amount, and turn half the length of the work. Throw out the feed, then stop the lathe, and, being careful not to touch the cross-feed handle, remove the work and run the tool back to the starting point. Change the dog to the finished end and keep the protecting piece of brass or copper under the screw. Oil the dead center, adjust the work between centers, and turn to meet the finished cut already made. *If the position of the cross slide has not been changed, the ends of the piece will measure the same; if the live center runs true, the two cuts will meet exactly; if the centers are in line the piece will be straight.*

111. Turning Duplicate Pieces.—When turning a number of duplicate pieces, set the tool for the first piece, cut the required length, throw out the feed, then stop the lathe, remove the piece, run the carriage back to the starting point, and put in the next piece. Measure each piece when it is taken from the lathe to make sure the setting is unchanged.

If the cut is of sufficient length to give the necessary time, put a dog on the next piece and oil the center so that it will be ready to put in the lathe when the other is taken out. "Time is money."

When the first cut is made on all the pieces, take the next operation or cut in the same way—piece by piece, and so on till all are roughed, then take the finishing cuts by the same method of procedure.

These directions apply in general to any lathe operation, and as a matter of fact to most machine-shop work involving more than one piece. Machining several pieces without chang-

ing the setting of the tool or the adjustment of the machine makes for speed and accuracy.

112. Filing in a Lathe.—While it is true that a very large percentage of the round work in the machine shop is now finish turned in the lathe, or ground to size in a grinding machine, it is yet important that the machinist shall know how to file and polish in the lathe as well as at the bench.

Very little work requires any filing, although it often happens that a few brushes with the file will save considerable time in turning. For example, if a special taper is turned nearly correct, a few strokes of the file will make it right much more quickly than it could be turned, or if a shaft is nearly exact a very little filing will make it correct. Sometimes it may be necessary to file and polish such work as a filleted corner, or rounded edge or end, or some special part of a machine such as a bushing or a handle or a pulley.

Filing in a lathe is accomplished by moving the file slowly on the revolving surface. The file mostly used for lathe work is the mill file.[1] This file is single cut and does not pin so readily as the double-cut files.

The mistake is usually made of having the work, especially cast-iron work, revolve too fast. A surface speed about double that of turning will usually be found sufficient.

Another mistake that the beginner often makes is to move the file across the work too rapidly, thereby filing it out of round. A long slow stroke will always give the best results. The file should not be lifted on the return stroke but the pressure should be relieved.

Another fault is to bear on too hard when filing round work. Remember that the surface of the file at any given time touches only a very small surface of the work and bearing on too hard will tend to file the piece out of round and also tend to pin the file and scratch the surface.

The beginner should be warned against the tendency to bring his left arm in contact with the revolving dog or face-plate or chuck. It often happens that in order to file a surface

[1] For description of files see page 305.

fairly close to the dog or chuck jaws, the operator must assume a most awkward and tiresome position in order to reach over the chuck or faceplate with his left arm. It is an excellent idea to learn to file "left handed" in a lathe. It is about as easy to learn to file left handed as right handed and when the knack is acquired, a habit is formed that will make for *increased efficiency* and *greater safety*. A number of manufacturing concerns require left-handed filing in lathe work for the sake of safety.

Keep the file pointed practically straight ahead but the stroke should be from right to left or from left to right, to avoid the tendency to file more from one portion than another. Crossing the strokes in this way will help also to keep the file clean.

A better appearance of the work will result if the file (especially when new) is chalked when used on cast iron and oiled or chalked when used on steel.

Hold the file with the palm of the hand against the end of the handle and the *thumb on top*. Cover the end of the file with the thumb of the other hand and curl the fingers under.

Many machinists leave too much for filing. With a fairly smooth cut, two thousandths is enough to leave for filing and polishing.

113. Polishing in a Lathe.—Most polishing in the lathe is done with emery cloth. Emery cloth may be obtained in various sizes of grains from fine to coarse. Usually polishing is done with fairly fine grain of emery cloth and the best finishes are obtained with a piece of emery cloth practically worn out. A better finish and a more lasting polish may be obtained by applying a reasonable amount of lard oil.

As far as the operation of polishing is concerned, the speed of the work can hardly be too great, but if a large piece is in the lathe and a high speed is wanted, care must be taken to have this work well balanced, and if it is on centers have plenty of oil in the tail center and do not adjust the tail center too tight.

Do not try to file or polish work with grooves or slots or holes in the surface. If it is necessary to polish such a piece in a lathe, plug these holes or grooves with hard wood.

Questions on Turning—I

1. What do you mean by "adjusting" the work between centers?
2. How do you make sure the dog does not bind in the slot of the faceplate?
3. Why it is necessary to have the live center running true? Name two methods of truing a live center.
4. Why must both centers be in line? How are the centers aligned approximately? How are they aligned accurately?
5. Why is it necessary to put oil in the dead-center hole? Why is the dead center hardened?
6. What occurs if the tool is not caught short?
7. What occurs if the tool is set below the center? If set too far above the center? What is the best way of determining when the tool is set right as regards the center?
8. What occurs if, when turning, the dog hits the tool rest? How does laxity in setting the tool often account for an accident of this kind?
9. How may the proper cutting speed be calculated?
10. What determines the roughing chip? The roughing feed?
11. What determines whether the finishing chip shall be $1/64$ in. or more than that?
12. Why does a finishing chip of $1/64$ in. give better results than a finishing chip of $2/1000$ in.?
13. How is the roughed diameter measured? When is the measurement made? Why?
14. Explain the method of using, and the value of the graduations on the cross-feed screw.
15. If it is advisable to oil the center during the cut, how is it done? Why not simply put oil on the outside?
16. If machine oil is not heavy enough what may be used instead?
17. What determines the proper feed for the finishing cut?
18. How is the finished diameter measured? When is the measurement made? Why?
19. When is a protecting piece under the dog screw necessary? Why is a piece of copper or brass better than sheet iron?
20. Describe in detail the method of turning duplicate pieces, and explain the advantages of this method of procedure.
21. Why is a mill file best for filing work in a lathe?
22. What causes "pinning" in a file?
23. What two mistakes are often made when learning to file in a lathe?
24. When it is practicable to finish round work by filing in a lathe?

SHOULDER TURNING

When turning to a shoulder it is assumed that the work on which the shoulder or shoulders must be made has been faced. It may or may not have been turned.

114. Definitions.—In Fig. 95, *a* represents a shoulder with a "filleted" corner, *b* with a "square" corner, *c* illustrates a shoulder with a "rounded" edge, *d* with a "chamfered" edge, and of course the last two terms apply as well to the ends of a piece as to the edges of a shoulder. To "break" an edge is to

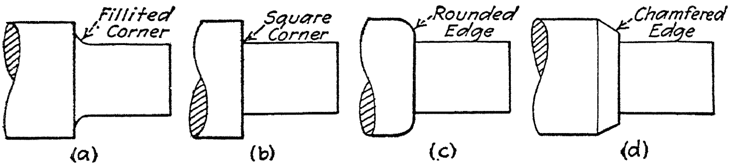


Fig. 95.

touch it lightly with a file or emery cloth and take away the extreme sharpness.

115. Roughing to the Shoulder.—When it is required to turn to a shoulder, first lay off the distance from the end. Use a hermaphrodite caliper, or a scale and scriber, and make a clean sharp line. Chalking the spot in which the line is to be scribed will serve to make it more distinct.

If more than one shoulder is to be cut on the end of a piece, it is a matter of *judgment* whether the shoulder nearest the end or farthest from the end should be turned first. The usual practice is to lay off the longest distance and rough turn to that shoulder, then lay off the shoulder nearer the end and rough turn to that, etc. This might involve unnecessary roughing cuts, however, if the shoulders were not very deep.

If there are several pieces, lay off the shoulder distance on several of them before shouldering any. Put the pieces between centers *without the dog* and revolve once or twice by hand and mark the line. Rough turn in the usual way, with the regular turning tool, until within 1/8 or 1/16 in. of the line, then throw out the power feed and feed *to the line* by hand.

It is well to grind the turning tool with a small round nose and to set it in such a way as to leave as little stock at the shoulder as possible for the finishing tool. When there are a number of pieces, rough turn all of them with the same setting of the tool.

116. Finishing the Diameter and the Square Shoulder.—On the longer shoulder cuts the regular turning tool used for roughing may be used also for finishing the diameter nearly to the fillet. Do not, however, turn quite to the fillet because then the last chip will be as heavy as the roughing chip and is likely to dig in and spoil the work.

The *shoulder tool* (Fig. 96) is used to rough out the fillet and also, at the same setting, to finish the diameter where the fillet was, and the square shoulder (on the shorter shoulder cuts it is used to finish the diameter its whole length). It is ground to have two cutting edges at right angles to each other and 1/16 in. or more wide; one edge to turn the diameter and the other edge to face the shoulder. Have the two cutting edges wide enough to give a good finish but not so wide as to tend to dig in. Extreme care must be taken when grinding and setting a square-shoulder tool. Then proceed as follows:

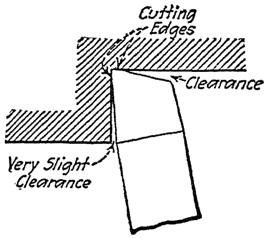


FIG. 96.

1. Using the square-shoulder tool, *with hand feed,* carefully cut out the fillet left in the corner by the roughing tool to leave the corner fairly square (taking two or more cuts to remove this material is called "stepping out" the fillet).
2. Take a cut on the diameter leaving it only 0.015 to 0.020 in. oversize, and note the setting (graduations) for this cut.
3. Face the shoulder to the line.
4. When the shoulder is faced to the line and the diameter is the small amount oversize, feed in to the graduation noted and take a light cut on the shoulder, *splitting* the line.

5. Feed in again to the graduation noted, and, in addition, *half* the number of thousandths the diameter is oversize, and turn the diameter to finish size.

NOTE.—The reason for noting the cross-feed setting is this: it may be necessary to take a second facing cut to get the right dimension and it will be easy to run the tool in exactly far enough and not risk undercutting the diameter. Also, if several pieces have been roughed, the finish cuts can be made quickly after the first setting because it has been determined how far in to run the tool. The smaller filleted corners are turned in the same way as explained above. The finishing tool is substantially the same except that it is rounded to give the required radius.

The reason for hand feed *always* when turning near a shoulder is the impossibility of throwing out the power feed at exactly the right instant. One-eighth inch is close enough to the shoulder to feed (long feed) by power.

The reason for leaving 0.015 to 0.020 in. on the diameter and splitting the shoulder line before turning the diameter to exact size is the less likelihood of "digging in" and undercutting the diameter near the shoulder.

Have the two cutting edges of the shoulder tool a trifle less than 90 deg. rather than more.

Many machinists prefer to face a shoulder by feeding in from the circumference. In this case the tool is set to split the line and fed towards the corner. In either case rough practically to the line; do not leave too much for the finishing cut or the corner of the finishing tool will become dull sooner than need be.

If possible a piece should usually be roughed all over before any finish cut is taken. Where the end is turned smaller to a shoulder it should be roughed first thereby possibly saving a second roughing cut over this portion of the piece, that is, a piece need not necessarily be roughed to a straight cylinder before roughing the shoulder cuts. After roughing, face to length. The next operation is to finish turn the larger diameters. Then lay off the exact shoulder dimensions on these finished diameters, finish the small diameters, and face the shoulders to the lines, checking the diameters with a micrometer and the lengths by scale measurement.

117. The Forming Tool.—The larger filleted corners are best roughed out with a large round-nose tool using both hand

feeds simultaneously. An approximately correct gauge cut from sheet metal may be advisable. The best tool to use for finishing a large fillet or a rounded corner of almost any size over $\frac{1}{8}$-in. radius is termed a forming tool. Forming tools are often forged and then filed to the desired shape. Figure 97 illustrates a forming-tool holder. In principle it resembles the old-fashioned "gooseneck" or "spring tool" in that any tendency of the tool to spring is *away* from the work and not into it. In the same figure are shown cuts of typical forming tools to be used with the holder. Usually a better finish is produced on steel if oil is used to lubricate

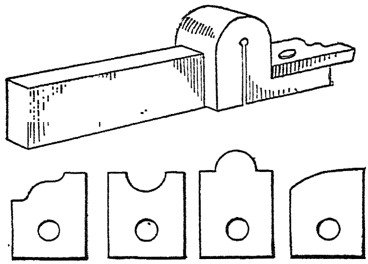


FIG. 97.

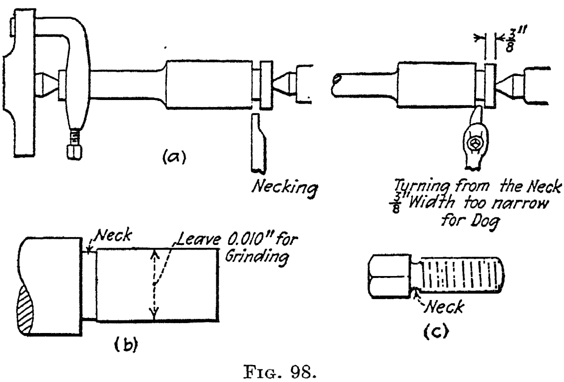


FIG. 98.

the forming tool. The cutting speed is fairly slow to avoid chattering.

118. Necking (Fig. 98).—Occasionally it is desirable when turning shouldered work, especially when roughing, to cut a groove or "neck" of nearly the correct depth and turn either to the groove or from the groove depending on which end of the piece it is most convenient to place the dog.

When a piece is afterward to be ground to a shoulder it is often advisable to neck it just under the size to be ground (*b*, Fig. 98) so that the grinding wheel will not have to be fed close to the shoulder. If a thread is to be cut to a shoulder it usually is considered better to cut a neck for the thread tool to run into (*c*, Fig. 98). The necking tool may be round nosed or square nosed, as desired.

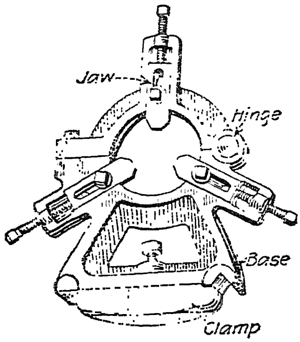


FIG. 99.

119. The Center Rest.—The center rest, or steady rest (Fig. 99), is a very valuable accessory to the lathe when slender pieces of any considerable length are to be turned. The base is planed to fit the inside ways of the lathe and by means of a clamp the

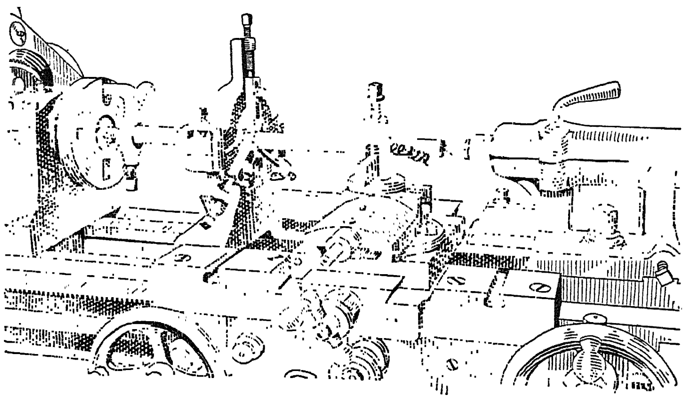


FIG. 100.

center rest may be held in the required position. The three jaws are adjustable.

A "spot" slightly wider than the jaws of the steady rest, is very carefully turned near the center of the work (see Fig. 100). It tne piece is to be turned the whole length and other opera-

tions afterward made on one or both ends, the machinist usually prefers the center rest to the follower rest (paragraph 120) for steadying the work during the first operation of turning the cylinder as well as for the later operations. In such a case the spot should be turned somewhat nearer the live center than the middle of the piece in order that the first cut may be turned at least half the length of the work. When one-half the length of the piece is turned, reverse the piece and readjust the jaws to the diameter of the turned part of

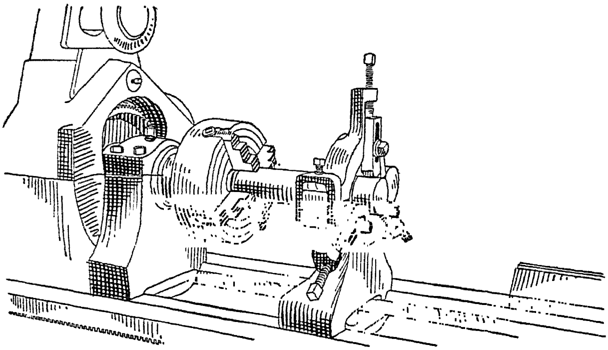


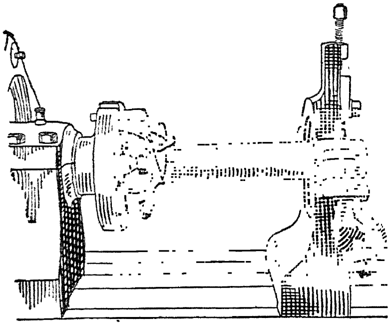
FIG. 101.

the work. To protect this part from being roughened a thin piece of brass or copper should be placed around the piece under the jaws; use oil to lubricate, and do not adjust the jaws too tight.

The spot should be turned smooth and preferably of some standard mandrel size. The reason for the particular diameter of the spot is to enable the operator to adjust the jaws on a short mandrel rather than on the piece itself, because a long slender piece springs so easily that it makes a true adjustment on such a piece very difficult.

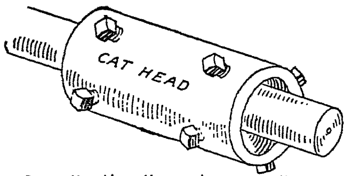
Do not adjust the jaws without first securely clamping the center rest to the ways, for unless this is done, the adjusting of the jaws is apt to lift it from the ways, and when it is finally clamped down the work is sprung.

A long piece of work placed in a chuck for the purpose of boring or turning may not be held rigidly enough, in which case, the center rest may be used as a support for the outer



end (Fig. 101). A spot for the jaws is first carefully turned concentric with the axis of the work.

If one end of the work is supported by the live center, drilling and boring may be done on the other end when the center rest is used as a support (see Fig. 102). In this case it is necessary to tie the work to the faceplate (usually with a belt lacing) in order to keep it tight against the live center. To tie the work to the faceplate, unscrew the faceplate three or four revolutions and tie the lacing as tight as possible. After thus tying, screw the faceplate home; this draws the work securely against the live center.



By adjusting the set screws the bushing (cat head) may be made to run true on bar of casting

FIG. 103.

The *cathead* (Fig. 103) may be used where it is impracticable to turn a "spot" for the center rest. Great care must be taken to adjust the cathead before putting it in the center rest so that it will run perfectly true when the work is revolving on centers.

120. The Follower Rest.—The follower rest (Fig. 104) is used to prevent slender work from springing away from the tool during the cut. The diameter is turned for a short distance to the desired size and the two jaws are adjusted to this diameter. As the follower rest is bolted to the carriage and

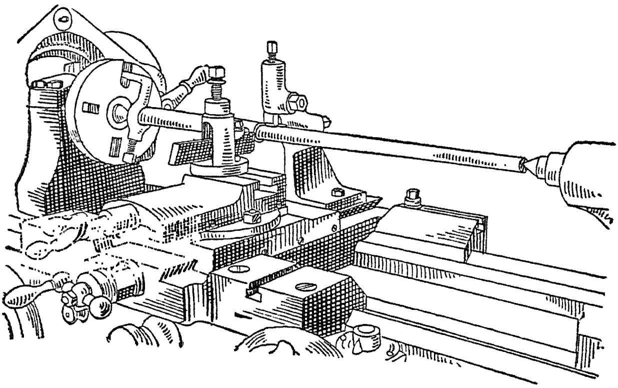


FIG. 104.—Using the follower rest.

moves with it, the two jaws constantly offer resistance to the spring of the work away from the tool.

121. Knurling.—Knurling may be defined as the process of checking the surface of a piece by rolling depressions into the surface. The knurled portion of a nut or of a screw that is to be adjusted by hand, or of a handle, as for example on a tap wrench, gives an excellent gripping surface.

Most knurled jobs done in machine-shop work are of the diamond pattern in "coarse," "medium," and "fine" as shown in Fig. 105. The knurls are small wheels with the marking cut in their faces and hardened.

The advantage of the particular kind of knurling tool shown in the figure lies in the fact that the wheels work opposite each other, and thus do not distort the center holes in the work. It is possible with this tool to knurl a piece of small diameter projecting from a chuck and this cannot be accomplished by a tool in which both knurls tend to push the work away from the tool.

Care must be taken in starting the knurl to see that one wheel does not "split the diamond." The knurls should be pressed hard into the work at the start and the pressure relieved somewhat after making sure they "track." The finished job should show as in the illustration with the diamond shape clean and sharp.

After a knurl is started right it will usually track right along at a medium or fairly coarse feed. If in the beginning

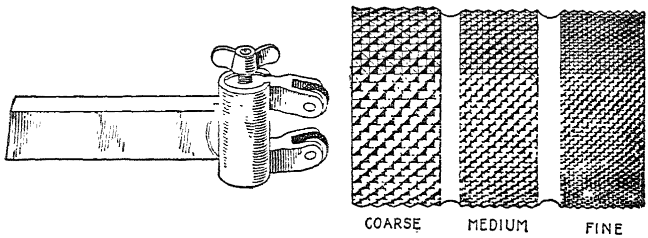


FIG. 105.—Knurling tool (*Billings and Spencer*) and illustrations (full size) of knurled work.

the wheels do not track but one of them splits the diamond, try another place on the work and then when the tool does make the proper knurl going over the split part will probably correct it.

Power feed may be used and usually one or possibly two "cuts" will be enough. Oil is used to lubricate when knurling any kind of material.

HOLDING WORK ON A MANDREL

122. The Standard Mandrel.—A mandrel (sometimes called "arbor") is a tool which when pressed into a finished hole in a piece of work provides centers on which the piece may be turned or otherwise machined.

A mandrel is a master tool and should be treated as such. Too frequently one sees a mandrel with the surface scored and cut, and the centers ruined. Such a tool is worse than useless, it had better be discarded altogether and not spoil more work. Furthermore, hundreds of jobs have been utterly ruined by reason of the operator not starting the

mandrel straight, or trying to force it, large end first, into the hole, or forgetting to put a little oil on the mandrel or

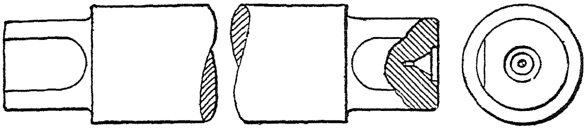


FIG. 106.

in the hole. Cultivate habits of carefulness, keep the tools in good shape, and make your work right.

Standard hardened and ground lathe mandrels (Fig. 106) are manufactured in various sizes. The mandrel tapers about 0.008 in. per foot, and the nominal size is near the middle. It is driven or pressed into the hole in the work and holds by friction. *To prevent damage to the work, the mandrel should always be oiled before being forced into the hole.*

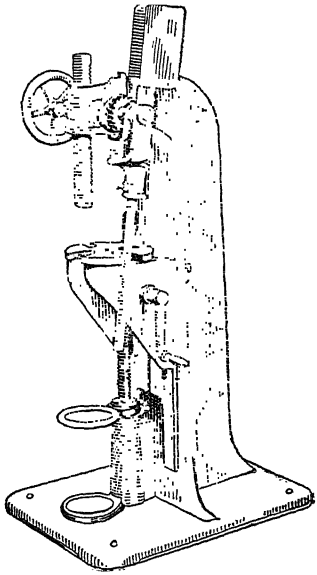


FIG. 107.—Mandrel press or "arbor press."

The ends are turned somewhat smaller than the body, so that any nicks or burrs, caused by the clamping of the dog, will not injure the accuracy of the mandrel.

It is very important to have the centers large enough to withstand the severe strain that may be caused by turning a heavy piece; that they are exactly 60 deg. to fit the lathe center; and very smooth to ensure a good bearing. The centers are recessed, that is, cut out a trifle around the center, for protection, but even so, a mandrel should never be driven with a steel hammer without protecting the end. It is better to use a babbitt hammer, or a press (see Fig. 107) made especially for forcing a

mandrel in or out. *Be sure the mandrel is started straight* or it will score one side of the hole. If the hole is properly sized the mandrel should enter a considerable distance before it begins to bind, thus serving to start itself square. It is important to remember that *the size of a mandrel is always marked on the large end.* The other end is, of course, the entering end. To avoid pressure against the arms or the web of a pulley when forcing a mandrel in or out put a collar under the hub; if a collar is not available a pair of short parallel pieces will answer.

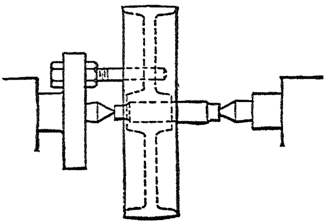


FIG. 108.

123. Using a Mandrel.—It is especially important when using a mandrel that the live center *runs exactly true.* Otherwise the mandrel will run out of true and the work will not be faced or turned true with the hole. Great care must be taken when adjusting a mandrel between centers, and also in attention to the dead center during the turning operation. If the mandrel gets hot, burns the oil out, and twists the center off, not only is the center damaged but the mandrel is ruined.

Turning and facing operations on work held on a mandrel are not different from other turning and facing operations, and require the same conditions of tool grinding and setting. When turning work on a mandrel feed toward the large end of the mandrel, if convenient. This tends to tighten the work on the mandrel. When turning comparatively large work on a small mandrel care must be taken that the mandrel does not spring or bend and also that the work does not turn on the mandrel. It is often advisable in such a case to drive the work directly from the faceplate instead of by means of a dog (see Fig. 108).

Always remember that, even if the mandrel is hard, the cutting tool is harder and it is easy, through carelessness, to injure the mandrel. Set the facing tool with a thickness of

paper between it and the mandrel and thus avoid cutting into the mandrel.

124. Other Forms of Mandrels.—*Home-made mandrels:* A machine-steel mandrel with case-hardened ends is often found efficient; or if it is a rush job a piece of machine steel carefully turned (on good-sized centers) to fit the hole will answer. On large sizes a satisfactory mandrel may be made of cast

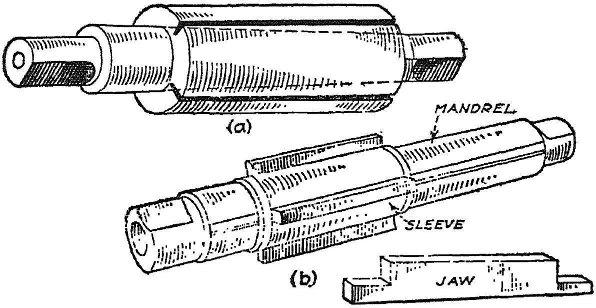


FIG. 109.—Two types of expansion mandrels.

iron by inserting in the ends hardened tool-steel plugs with centers.

Expansion mandrels of various types are manufactured. These mandrels are expensive and are not so accurate as the solid ground mandrel. *Taper mandrels* with cast-iron expansion bushings, (*a*) Fig. 109, are extensively used, especially in the larger sizes. Do not try to force the bushing to bind in a hole that is oversize or it is likely to ruin the bushing.

Another form of expansion mandrel is shown in (*b*) Fig. 109. It consists of the mandrel proper which has four or more grooves cut lengthwise uniformly deeper toward one end; a sleeve with slots opposite the grooves in the mandrel and of the same width; and the jaws which fit into the grooves of the mandrel and through the slots in the sleeve. The jaws taper in length to correspond to the incline of the slots and therefore bear evenly in a straight hole. The take-up or release is made by sliding the mandrel in the sleeve. The particular value of this mandrel lies in the fact that it may be

adjusted to bind in a hole somewhat larger or smaller than its nominal size.

A *gang mandrel* (Fig. 110) is useful for turning or milling several pieces such as gear blanks or cutter blanks at the same time. It is especially useful when turning thin pieces.

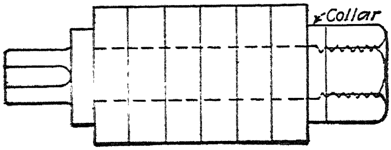


FIG. 110.

The threaded mandrel or "nut arbor" (Fig. 111) is used for facing nuts or otherwise machining inside threaded pieces. Unless one face of the work is square with the axis of the thread

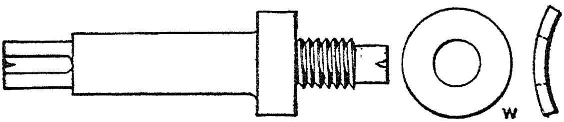


FIG. 111.

an equalizing washer *W* is necessary to make sure that the nut takes up true on the thread of the mandrel and is not canted. Such a washer is quickly made from an ordinary iron washer by bending it slightly.

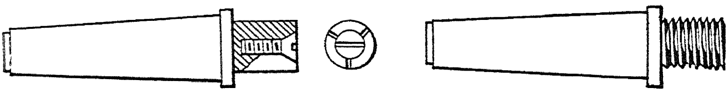


FIG. 112.

The taper-shank mandrel (Fig. 112) may be fitted to the taper hole in the spindle of the machine. The projecting portion may be of the form desired. A nut mandrel made in this way is used for machining "blind nuts," that is, a nut in which the hole does not go through.

125. To Turn a Crankshaft or an Eccentric.—When a cylindrically turned surface of a piece has an axis parallel to, but not coincident with, the normal axis of the piece, this surface

is said to be turned eccentric. The work itself may be called an "eccentric" (*a*) or a "crankshaft" (*b*) and (*c*), Fig. 113. Both are much used in machine construction to convert rotary into reciprocal motion or the contrary. To turn the eccentric surface it is necessary to provide centers offset from the centers of the normal axis an amount equal to one-half the

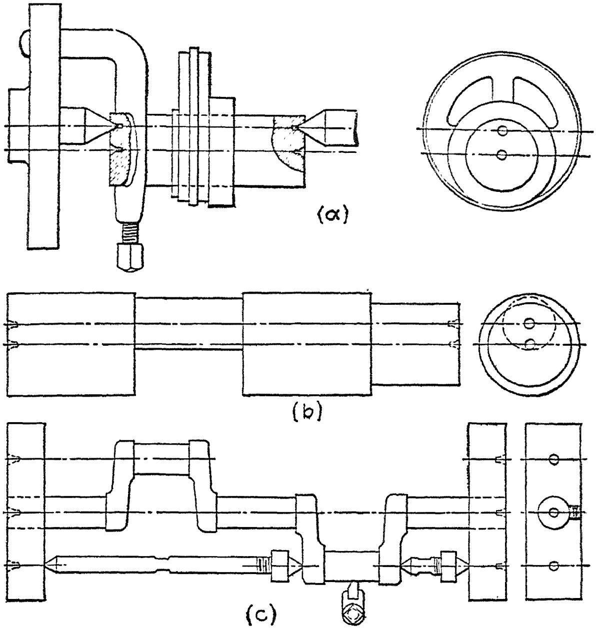


FIG. 113.

"throw" desired, that is, one-half the amount of the reciprocating motion to be imparted or converted as the case may be. Three methods of providing the offset centers for turning eccentric cylindrical surfaces are illustrated in Fig. 113. An eccentric (*a*) is usually keyed to a shaft when in use and consequently is located on a mandrel, by a key or by a witness mark, and then turned. A mandrel with offset centers is shown in (*a*). The crankshaft (*b*) is itself provided with an extra pair of centers. When the amount of eccentricity is too great to

allow the extra pair of centers within the diameter of the normal bearing, special pieces with centers properly arranged may be fitted on the ends of the shaft as shown in (*c*). A suitable counterweight should be provided, and also braces to eliminate spring.

Questions on Turning—II

1. If considerable metal is to be removed, is it advisable to face to exact length? Give reason.

2. Why is it usually advisable to lay off the shoulder distance from one end of the work?

3. What tools may be used to lay off these distances? When is chalk used? When is blue vitriol used?

4. Is care used in laying off shoulder distances when roughing? Why?

5. If you are going to rough a square shoulder, how does the shape of the tool differ from a regular turning tool? How is the tool set? Why?

6. If one or more diameters are to be turned on a piece, is it necessary to rough to straight cylindrical shape before roughing the smaller diameters?

7. When roughing out how much stock do you try to leave for finish facing the shoulder? Why?

8. Why is it good practice to throw out the power feed when 1/16 in. or more from the shoulder, and feed the rest by hand?

9. When turning how much do you leave for a finishing chip?

10. How do you grind a combination turning and facing tool for finish turning the diameter and finish facing the shoulder? Is such a tool efficient? Why? Explain its action in detail.

11. Explain how the graduations on the cross-feed screw may be of great help in turning shouldered pieces.

12. Sometimes when turning shoulders it is convenient to neck the work. How is this done?

13. If the work, when faced, is left somewhat overlength because of the amount of stock to be turned off, when should it be faced to length? Why at this time?

14. What is meant by a square corner when turning a shoulder? A filleted corner?

15. What is a square edge? A rounded edge? A chamfered edge?

16. Why must care be taken when starting the knurling tool?

17. How do you tell when a piece is sufficiently knurled?

18. Why are two wheels used to produce the diamond-pattern knurl?

19. What is meant by turning a "spot" for the steady rest?

20. Why is it advisable to make the spot a nominal size if convenient?

21. Why not turn the spot in the middle of the piece?

22. When is it necessary to tie the dog to the faceplate? How is it done?

23. What is the difference between a steady rest and a follower rest?

24. Why is a mandrel tapered slightly?

25. Why is a mandrel not marked on both ends?

26. How is a mandrel started and how is it pressed in the hole?

27. What precaution must always be taken before forcing a mandrel in a hole? Why?

28. Name three kinds of mandrels and explain the particular value of each.

29. What is a "nut arbor"? What do you understand to be the value of the bent washer on the nut arbor?

30. What do you mean by a blind nut? Can you find a blind nut holding any part of the lathe?

31. How may a blind nut be held while being faced or turned?

32. A special bushing 3 in. long has a hole 0.990 in. in diameter. What kind of a mandrel, if available, could be used?

33. If no mandrel for a hole 0.990 in. in diameter were available, what would you do?

34. Why should the center holes in a mandrel be clean cut, smooth, and fairly large?

35. Why are the ends of a mandrel recessed around the centers?

36. Explain how work of fairly large diameter, mounted on a mandrel, may often be driven direct from the faceplate. What is the advantage of this method of driving?

37. What is an eccentric?

38. What do you understand as the difference between an eccentric and a crankshaft?

39. What do you mean by turning eccentrically?

40. Using one or more V blocks and a surface gauge how would you proceed to lay out the positions of the two pairs of centers in (*b*) Fig. 113?

CHAPTER VIII

CHUCKING WORK

Engine-lathe work may be divided into three general classes; work on centers, chucking work, and faceplate work. Each of these classes of work may involve one or all of the operations of turning, threading, boring, etc. Certain operations relative

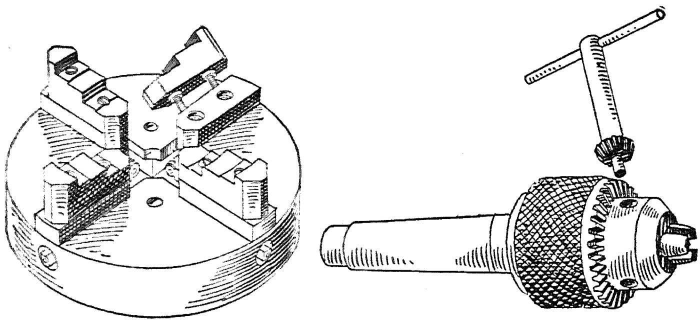


FIG. 114.—Lathe chuck and drill chuck.

to work on centers have been explained and chucking work will now be considered.

126. Kinds of Chucks.—The larger chucks (Fig. 114), those that will hold a piece of a diameter of 2 in. or over, are usually mounted on a chuck plate that screws on the end of or is fastened to a flange on the machine spindle. In lathe work they are called "lathe chucks." The smaller chucks are provided with a taper-shank arbor which fits the taper hole in the end of the spindle (see Fig. 114). Because their chief function is that of holding small drills, etc., these chucks are generally called "drill chucks."

If the jaws move independently of each other the chuck is called an "independent chuck"; if they all move together,

it is called a "universal chuck." A chuck that may be arranged as either independent or universal is called a "combination chuck." Practically all of the larger chucks are independent or combination while the small lathe chucks and

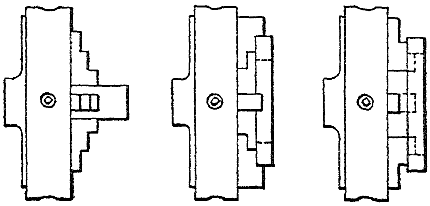


FIG. 115.—Illustrating three ways of holding work in a chuck. In (*a*) and (*b*) the work is held by gripping it on the outside; in (*c*) the jaws grip the surface of the hole with an outward pressure. It will be noted that such a piece of work as the ring illustrated may be faced on one side and bored when held as in (*b*), and the other side faced and the outside turned when held as in (*c*).

the drill chucks are universal. Most lathe chucks have reversible jaws. In some chucks the jaw is removed and reversed and in other designs (see Fig. 114), a part of the jaw is reversed. The object of the stepped reversible jaw is to

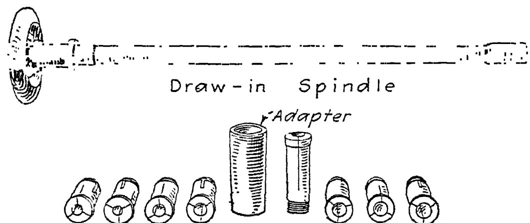


FIG. 116.

enable pieces of various diameters to be held. The work may be gripped either inside or outside as desired (see Fig. 115).

The spring chucks (Fig. 116) are very handy for small round work. The adapter fits the taper hole in the main spindle and the chuck fits into the adapter. The hollow draw-in spindle goes through the hole in the lathe spindle, and the

end, being threaded inside screws over the threaded portion of the chuck and draws the chuck back into the adapter until it grips the work. This form of chuck is often called a spring collet or a draw-in chuck.

127. Chuck Work.—In such work as gear blanks, pulleys, collars, bushings, etc., where the outside must run true with the hole, the hole is finished first, usually in a lathe chuck. A reasonable amount of stock is allowed on the length and diameter and later the piece is faced, turned, and possibly other-

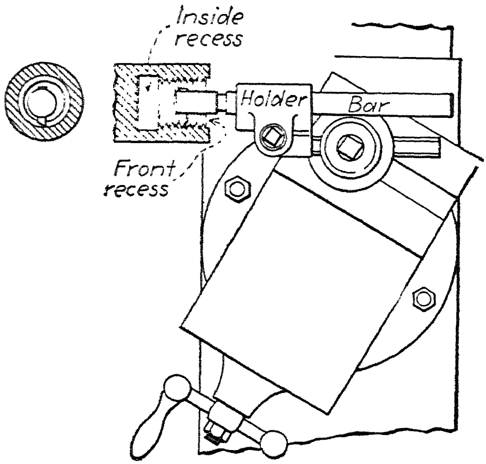


FIG. 117.—Boring a thread.

wise machined while being held on a mandrel (see page 152). Many times the job must be drilled, bored, reamed, recessed, and possibly threaded, while being held in the chuck. A chuck job with several operations is shown in Fig. 117.

The lathe chuck is frequently used, also, for holding a bar to be turned, shouldered, threaded, etc., on the end. So long as the hole through the main spindle is large enough to permit the bar to go through, any kind of lathe work may be done on the end that projects. Such a job is shown in Fig. 118. The machined end may or may not be cut off as occasion requires. The cutting-off operation is shown in Fig. 122.

Fairly large pieces are easily held in a chuck and faced (see Fig. 120). For ordinary facing of jobs in a chuck, see page 125.

128. Selecting a Chuck.—When selecting a chuck remember there is no need for a large chuck (12 in.) if a small one (6 in.) will do. There is no need of taking the trouble of truing work in an independent chuck if a universal chuck is true enough. A chuck with No. 2 jaws (*a*, Fig. 115, p. 161) should be used

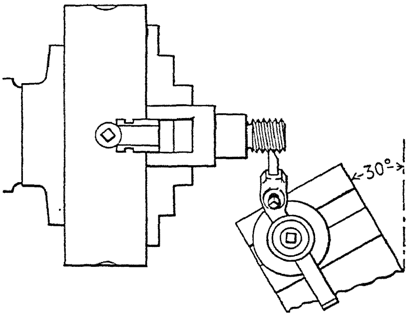


Fig. 118.

unless the work is so large in diameter as to require No. 1 jaws (*b*, Fig. 115).

The lathe spindle sizes are not yet standardized and chucks are not interchangeable on various lathes. Get the chuck that fits the particular lathe.

129. Removing the Screw-on Faceplate or Chuck.—Have a substantial block of wood, say 2 in. square, long enough to rest on the back ways and reach the slot in the faceplate when the slot is in a position corresponding to "three o'clock." Arrange the lathe for a fairly slow speed and running it backwards (usually by hand) allow the edge of the slot to strike sharply against the end of the block. This will jar the plate loose. A chuck is removed in the same way by bringing one of the jaws against the block.

Caution.—A chuck or faceplate should never be run off or on a spindle by machine power. Either may sometimes be *started* off, for one or two threads, but *never started on.*

130. Mounting the Chuck on the Spindle.—Remove the live center and if short work is to be machined put waste in the center hole to keep out chips and dirt. Remove the lathe faceplate and place it where chips and dirt will not get in the threaded hole. Be sure the thread on the spindle, and also the shoulder of the spindle, are perfectly clean and free from any burr; likewise the chuck plate. Oil the thread thoroughly. A chuck 10 in. in diameter or over should be brought in line with the spindle by placing it on a suitable block laid across the ways of the lathe; start it square on the thread, turning the spindle by hand (the same care should be taken when removing the chuck). The chuck should screw on the spindle easily. It should be screwed tight against the shoulder by hand but must *not* be brought up with a bang.

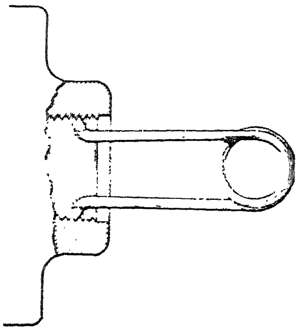


FIG. 119.—Spring cleaner for chuck plate or faceplate. Made of drill rod or hard brass. May be used for various sizes.

131. Adjusting the Work in an Independent Chuck.—Adjust the jaws to about the size of the piece to be held and approximately central with the concentric grooves turned in the face of the chuck. Tighten the jaws on the work, and turn the chuck around by hand to be sure it will revolve without trouble.

Put a toolholder in the tool post, butt end against the work, and make it merely finger tight—no wrench Now revolve the work and the "high spot" will just touch the toolholder. Mark the high spot in some manner, and turn the chuck around halfway. Suppose there is, in this position, 1/16 in. space between the work and the toolholder, then loosen the nearest chuck jaw and tighten the opposite jaw to move the work just half the sixteenth, thus pushing the work to center. Then test again.

Another method of truing the work is to mark the high spot with chalk as the work revolves, stop the lathe, and true

the work, but this means guessing at the amount to move the work.

Tighten all jaws evenly and securely after the work runs to suit. Mark two adjacent jaws, and for subsequent pieces of the same size to be handled, use only these two jaws. If a three-jaw chuck is used, one jaw only need be loosened to release the piece. This will save time. Use judgment when tightening any piece in a chuck. If it is thin or weak it may spring or break if too much tightened. Sometimes a piece with considerable metal removed in boring, etc., gets thin and the jaws have to be eased up before the finishing cuts.

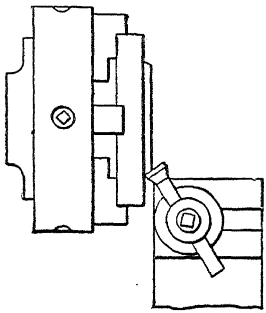


FIG. 120.—Radial facing.

If it is required to have the work "trued up by" a finished surface (outside, inside, or faced surface) an indicator may be used to obtain a fairly accurate result. It is, however, practically impossible to make it run exactly true. For this reason, if part of an operation is *finished,* do not remove the work from the chuck, unless it is absolutely necessary, until the whole operation is finished. There are times, however, when it is necessary to true up work to make a previously made hole run true. If a mandrel to fit the hole is available, put it in the hole (not tight) and true up the mandrel by indicator or otherwise. When the mandrel runs true, the hole is true. Then pull out the mandrel and proceed with the job. This method is usually easier than "indicating" the hole.

132. Radial Facing.—The term "radial facing" may be applied to the truing of the faces of work of a comparatively large diameter when held in a chuck or on a faceplate (see Fig. 120). The shovel-nose tool (see page 77) may be used for the roughing cut. When using this sort of tool or a similar tool, face from the circumference of the work toward the center. When finishing, an ordinary turning tool, or side tool, whichever is preferred, may be used. In radial facing care must be

taken that there is little or no end motion in the main spindle bearings. Also, to prevent the tool working away from the cut, and thus producing a surface which is not flat and true, it is necessary to tighten the carriage clamping screw. A much better finished appearance may be obtained on cast iron if the chip is very light and the feed is very coarse. A square-nosed tool makes an excellent cast-iron finishing tool. It is good practice when facing steel in a lathe to apply a suitable amount of lard oil or other cutting compound with a brush. Cast iron and brass are machined dry.

CUTTING OFF BAR STOCK IN A LATHE

133. The Cutting-off Tool.—Select an offset cutting-off tool (sometimes called a parting tool) with the blade only a trifle longer than half the diameter of the stock.

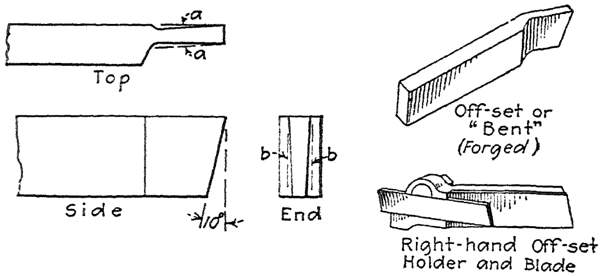


FIG. 121.—The cutting-off tool. The amount of clearance at (*a*) and also at (*b*) is very small, just enough to clear the sides of the groove being cut. Excessive clearance weakens the tool. One particular advantage of the patent cutting-off toolholder and blade is that the blade may be adjusted to project only the distance necessary.

The cutting edge of a cutting-off tool (Fig. 121) is the widest part of the blade. The front clearance is the same as for any turning tool, about 10 deg. Besides the front clearance it has clearance on both sides, toward the bottom, and also toward the body. The width of the cutting edge depends on various conditions but should always be wide enough to permit of the blade being strong, and not so wide as to unnecessarily waste the stock. A cutting-off tool never has side rake and seldom if ever any front rake. The reason for no front rake

is the decided tendency for the tool to "hook into the work" due to the slack in the cross-feed screw. To sharpen a cutting-off tool, grind it on the front.

134. The Cutting-off Operation.—Set the tool on center; if too high it will "ride" as it approaches the small diameter, if too low it will "dig." If the work is steel or wrought metal, use plenty of lard oil or some good cutting compound, and provide means of catching the surplus thus keeping the lathe clean. Feed by hand and be sensitive to the cutting conditions. The lathe, especially if motor driven, may be speeded

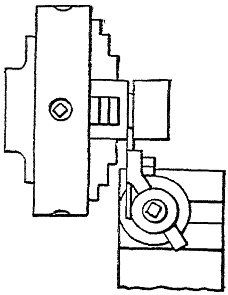


FIG. 122.—Correct use of cutting-off tool.

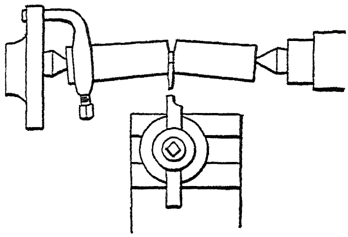


FIG. 123.—A sure way of spoiling the work and breaking the tool.

faster as the diameter is decreased. It requires strict attention to business to do a good job in grinding, setting, and using a cutting-off tool. When cutting off stock held in a chuck it is best to use a bent tool to permit of working close to the chuck (Fig. 122).

It should be emphasized here that spring of the work or of the tool is always a decided disadvantage. When using the cutting-off tool particular care should be taken that the work and the tool both be as rigid as the nature of the job will permit.

Do not attempt to cut in two a piece of work held between centers, do not attempt to even neck the piece if it is slender, because it will almost certainly be bent and ruined and the tool will be broken (Fig. 123).

135. Chattering.—The rapid vibration of the tool and the work which is called chattering frequently takes place when

using a cutting-off tool and may be due to one or more of several reasons: a tendency of the tool or work to spring; to the fact that the tool is set too high; to the looseness of the cross slide; or to the looseness of the lathe spindle in its bearings.

Questions on the Use of Lathe Chucks

1. How is the chuck started on the thread? If it is a heavy chuck how is it best held when starting?

2. What is the danger of having the spindle revolve by power when starting the chuck on the thread? When screwing it home?

3. Why should the chuck not be screwed against the shoulder with a bang? Why should it be screwed up tight?

4. To make the work run more nearly true, shall you adjust two opposite jaws to push the high spot *toward* the center or *away* from the center? Why?

5. If the high spot should come between two jaws, how would these jaws be adjusted?

6. What is the purpose of the rings in the face of the chuck?

7. If you had several pieces of the same size to machine in the chuck explain how you could save time by marking two jaws of the chuck.

8. Find as many of the following chucks as are available: 6-in. three-jaw universal; 6-in. two-jaw universal; 8-in. four-jaw independent; and a combination chuck.

9. What is the advantage of the universal chuck?

10. What advantages has the independent chuck?

11. How is the combination chuck changed from independent to universal? How are the jaws adjusted, to make them true, before the change is made?

12. There is a proper method of removing a chuck from the lathe spindle. How is it started? How is it held when being unscrewed? What is the danger of starting the lathe by power?

13. What type of chuck jaw is used to hold bar stock or pieces of small diameter?

14. How would you hold in a 6-in. chuck a piece 6 in. in diameter to bore it? How would you hold it to turn off the circumference?

15. What is meant by reversible jaws?

16. What is meant by jaws with reversible tops?

17. How do you size up the workman who hammers the chuck wrench or who uses a pipe extension to the wrench?

18. What is the danger of stopping the chuck by clapping the hand on it?

19. What is the small chuck with a taper arbor which fits into the tail spindle usually called?

20. What are two advantages of the split chuck (spring collet)?

21. Explain in detail the action of the draw-in sleeve in closing the split chuck?

22. What is the advantage of an offset cutting-off tool in chuck work?

23. How many clearance angles has a cutting-off tool?

24. When is a cutting-off tool set properly?

25. What is the objection to holding work between centers to cut in two?

26. What is meant by chattering? How may it be prevented?

27. When radial facing, what tool may be used for roughing?

28. What is the object of clamping the carriage when radial facing?

DRILLING AND REAMING IN A LATHE

136. Introduction.—*Drilling* a hole may be defined as one process of making a hole where none existed previously.

Boring may be differentiated from drilling in that it is the process of enlarging, by turning inside with some form of boring tool, a hole already existing, for example a hole already drilled, or a cored hole in a casting.

Reaming is the process of finishing a hole to the required size, by means of either a machine reamer or a hand reamer.

Many holes do not have to be especially accurate and merely drilling the hole may be enough. Or it may be the drilled hole is not accurate and smooth enough but one that is drilled $1/64$ or $1/32$ in. undersize and then finished with a machine reamer is all that is desired. Or possibly a hole drilled, say, $1/32$ in. undersize, and machine reamed with a machine reamer $3/1000$ to $5/1000$ in. under, and then finished with a hand reamer will be required. However, none of these holes have to be bored. The reasons for boring a hole are given at page 187.

While the drill press is essentially the machine for drilling and reaming, it often happens that it is more profitable to finish the holes in a lathe.

Before explanations of the setups for drilling and reaming are given, it is perhaps best to discuss briefly the tools used. So many drilling jobs are done in the lathe that it is necessary, in the beginning of one's lathe practice to know about drills,

how to sharpen and how to use them. Also a knowledge of the kinds and uses of reamers is important.

The twist drills and reamers used in lathe work are the same as those used in drill-press work. The flat drill though considered old fashioned by some is however a very efficient tool for drilling in a lathe.

137. Flat Drill.—Figure 124 illustrates the flat drill, and the "holder" which is clamped in the tool post. The two

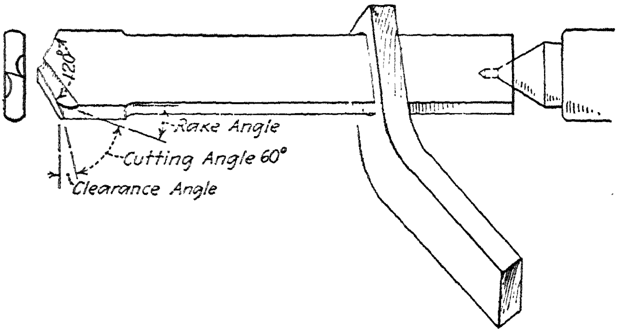


FIG. 124.—Homemade flat drill and holder.

cutting edges of the drill (called the lips) are at an angle of 120 deg. to each other, and are of equal length, which keeps the point central and the strain equalized thereby drilling a hole approximately the size of the drill. Note the *clearance angle* of the lip, also the *rake angle.* The *cutting angle* of the lips is about 60 deg. which is correct for metal cutting.

138. The Twist Drill.—The twist drill (Fig. 125) has almost entirely superseded the flat drill. The spiral flute forms the *rake angle* and gives a cutting angle of about 60 deg. Examine a twist drill and note that after the flutes are cut the remaining portion of the surface of the body is backed off or relieved and only a very narrow "land" is left. This body clearance reduces the friction of the drill in the hole.

Twist drills are made in number sizes, No. 1 (0.228 in. diameter) to No. 80 (0.0135 in. diameter). They are made in letter sizes A (0.234 in. diameter) to Z (0.413 in. diameter)

(see page 407). They are also made in sizes ranging by sixty-fourths of an inch from $\frac{1}{64}$ to 2 in. or more, and in metric sizes, ranging from 0.4 mm. (0.0157 in.) to 50 mm. (1.068 in.) by 0.1 mm. (about 0.004 in.) on the smaller sizes, and by

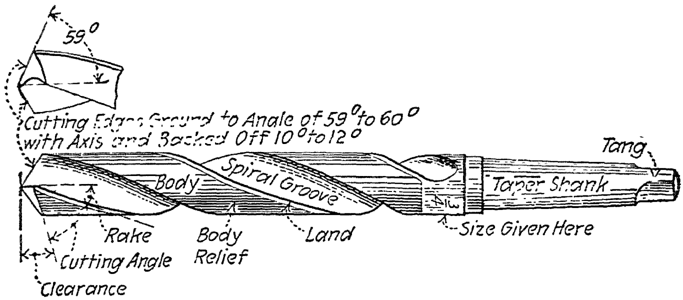


FIG. 125.—Taper-shank twist drill.

0.5 mm. on the larger sizes. The smaller drills are not marked and the size is found by the use of a drill gauge (Fig. 126).

Straight and Taper Shanks.—Drills are made with either straight shanks or taper shanks (Morse tapers). It is not

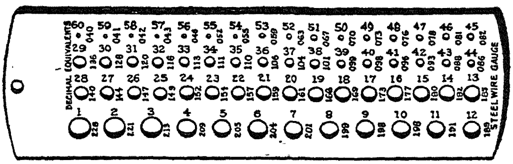


FIG. 126.—Drill gauge. The holes are used to gauge the sizes of drills which are too small to permit of having the size marked on them. The number and corresponding size of the drill are shown beside the hole. The "Jobbers Drill Gauge," which is similar to the gauge illustrated, shows the sizes from $\frac{1}{16}$ to $\frac{1}{2}$ in. varying by 64ths.

usually considered economical to buy taper-shank drills of a size smaller than $\frac{3}{8}$ in. diameter because of the added cost of the shank. Small drills are better held in a chuck. In the larger sizes (over $\frac{1}{2}$ in.) the difference in price between straight and taper shanks is not noticeable and it is nearly always more convenient to hold them by means of the taper shank.

139. Sharpening a Drill.—The clearance angle on a drill is about 12 deg. at the cutting edge (see Fig. 127). If correctly

sharpened, the angle of the edge across the web of the drill (the "dead center" of the drill) will be about 45 deg. with the line of the cutting edges (see *a*, Fig. 127). The appearance

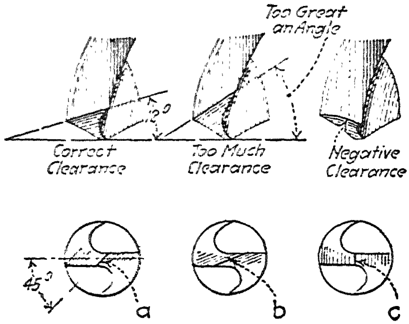


FIG. 127.

of the dead-center edge is an index to the clearance; when it is like *b*, the lip has too much clearance and when it is like *c* the lip has no clearance. It is very important that a drill should have sufficient lip clearance as it takes considerable

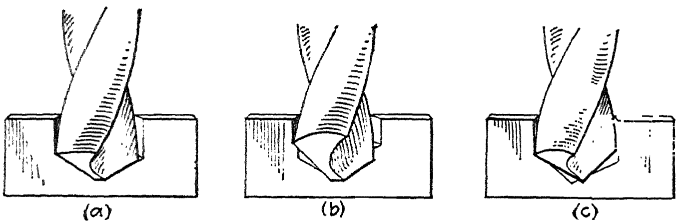


FIG. 128.—(*a*) Lips of different lengths. Drill will cut over size. (*b*) Lips with different angles with the axis of the drill. Drill will wobble and cut oversize. (*c*) Lips with different inclinations. One lip does practically all the work which tends to crowd the drill toward the opposite side and wear off the land.

pressure to feed the drill into the work under the best possible conditions, owing to the nature of the point, and if the lips are not properly backed off the drill will break under feeding pressure simply because it cannot cut.

Extreme care must be taken to get the lips exactly the same length and both at the same angle with the axis of the drill or the hole will be oversize. This is illustrated in Fig. 128.

Theoretically if the drill is ground at an angle of 59 deg. with the axis, a straight lip will be the result. Grinding the drill at any other angle with the axis results in a slightly curved lip. Therefore the drill should be held at about 59 or 60 deg. with the face of the grinding wheel as shown in Fig. 129. The observant student will soon learn to notice any inequality in the lengths of the lips or in their angles with the axis of the drill.

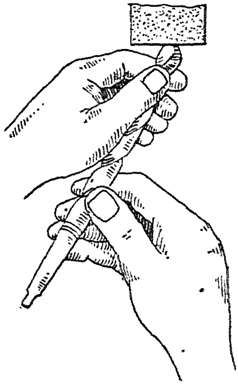


Fig. 129.

When sharpening the drill place the left hand on the tool rest, and hold the drill at 60 deg. as shown in Fig. 129. Hold the cutting edge to be ground in plain sight and in a horizontal position. In order to give 12-deg. clearance to the cutting edge hold the *shank* end of the drill down a little, that is, the right hand is a little lower than the left hand. Now bring the cutting edge against the grinding wheel with a slight pressure and at the same time lower the right hand a little farther. Carefully moving the drill against the grinding wheel a few times in this way will sharpen the edge and give the necessary clearance. There follows a summary:

1. Do not rest the drill on the tool rest, but rest it in your left hand, and your hand on the tool rest.
2. The axis of the drill should be 60 deg. with the face of the grinding wheel, not 90 deg. or 45 deg.
3. The cutting edge should be held *up*, where you can see it, not upside down.
4. Keep the cutting edge horizontal. Do not twist the drill when grinding.
5. Watch the *clearance*. Have about 12 deg., not twice that much, and certainly not a *negative* clearance angle.
6. Grind slowly and carefully. *Know* what you are doing.
7. Grind both lips alike.

It is suggested that in order not to waste the drill the beginner practice on a piece of flat stock, say 1/8 by 1 in., to grind it like a flat drill as shown in Fig. 124. When he has the knack of grinding a flat drill easily and quickly he will have less difficulty in learning to sharpen a twist drill.

To strengthen a drill the web is made thicker toward the shank. This is not noticeable on drills under 3/4 in. diameter

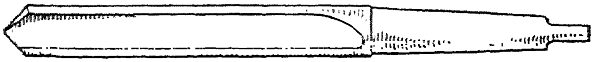


FIG. 130.—Taper-shank straight-fluted drill.

but on larger sizes, as the drill is shortened, it becomes necessary to grind the point somewhat thinner. Use a fairly thin round-faced grinding wheel, that is, somewhat smaller than the groove in the drill, and take care to grind an equal amount from each side.

For drilling brass, the lips of the drill should have no rake, and a straight-fluted "Farmer" drill (Fig. 130) is the best to use, but a twist drill may be used if the front of each lip is ground as shown in Fig. 131. It is also advisable to use a drill without rake when drilling very thin pieces owing to the tendency of the drill to "hook into" the work when it is breaking through.

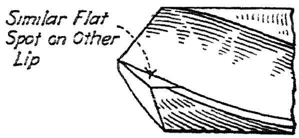


FIG. 131.

It often happens through ignorance or carelessness that a drill is used after it becomes dull, which causes the land to become worn away for a distance back from the cutting edge. The diameter of the end of the drill is reduced, thus making the drill bind and squeak. It will be necessary to grind off the undersize part and then sharpen the drill. *Always examine a drill before using it.*

140. Drill-grinding Machine (Fig. 132).—In shops where any considerable amount of drilling is done it is economical to have a drill-grinding machine. This machine may be quickly adjusted to support a drill of any length or diameter in a wide range of sizes and is so designed that it is a very simple matter

to grind the drill properly, that is, with lips of equal length, at the correct angle with the axis, and with the correct clearance.

141. Speeds and Feeds of Twist Drills—Cutting Compounds.—Owing to the variations of the hardness and toughness of the materials used in machine-shop practice, no hard and fast rule can be given for the speeds and feeds of twist drills.

The correct speeds and feeds must be determined by the judgment of the operator, and the following hints will help the beginner to obtain this necessary knowledge.

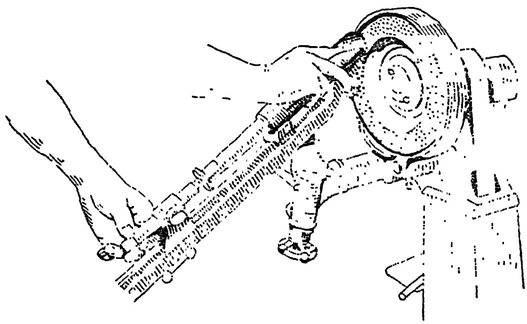


FIG. 132.—Drill grinder.

When the lip breaks off the feed is too heavy, or the drill has been given too much clearance as shown in *b*, Fig. 127.

When the drill splits, there is too much feed or the drill has not been given enough clearance. There seems to be a tendency for the beginner to give insufficient lip clearance toward the center of the drill. The whole length of the lip must be backed off or the drill will surely break under the feeding pressure.

The rapid dulling of the drill especially at the outer ends of the lips (the corners) is evidence of too much speed.

When a drill squeaks it is usually an indication of a crooked hole or dullness. Never allow a drill to squeak.

The following tables of feeds and speeds are here given as having proved practical for average conditions:

Feeds for Drills (average):

0.004 in. per revolution for drill 1/4 in. diameter to 0.015 in. per revolution for drills 1 1/2 in. and larger.

Same feeds for high-speed and carbon-steel drills.

Speeds of High-speed Steel Drills (average):

50 ft. per minute for annealed carbon steel.

70 ft. per minute for soft steel or cast iron.

200 ft. per minute for brass.

About half the above are safe speeds for carbon-steel drills.

Cutting Compounds Used in Drilling:

Unannealed steel—turpentine.

Carbon steel—lard oil or soluble oil cutting compound.

Soft steel or wrought iron—lard oil or soluble oil.

Malleable iron—soluble oil.

Brass—dry, or lard oil and kerosene mixture.

Aluminum, copper, and other soft alloys—kerosene.

Cast iron—dry, never use any cutting compound when drilling cast iron.

Questions on Drills and Drill Grinding

1. How is the drill held when grinding by hand—is it placed on the hand rest, or held in the left hand? Why?

2. How is the drill grasped with the right hand?

3. What angle does the center line of the drill as properly held make with the face of the grinding wheel? Why not 45 deg.? Why not 30 deg.?

4. Why is the drill held with the cutting edge up and in a horizontal position?

5. How is the drill moved against the wheel to "back off" the cutting edge? Why not give it a twisting motion?

6. What do you mean by fulcruming the drill in the left hand?

7. Why must care be taken to have plenty of water available when grinding a drill?

8. If a properly ground drill is held perpendicular to a flat surface, what angle will the cutting edge make with the surface?

9. How much clearance has the cutting edge of the drill?

10. What part of the twist drill is the lip? The point? The land?

11. What is the effect of too much lip clearance? Of not enough lip clearance?

12. How can you tell by looking at the point of a drill whether or not the drill has been given sufficient lip clearance?

13. Has the drill any other clearance?

14. What is "rake" on any cutting tool?

15. What governs the amount of rake angle on a twist drill?

16. Why cannot a set rule be given for the speeds and feeds of drills?

17. What does a squeak indicate in drilling?

18. What do you mean by the land of the drill being worn away? What causes this?

19. How many r.p.m. should a $\frac{3}{4}$-in. drill be run to give a cutting speed of 35 ft. per minute?

142. The Reamer.—It is practically impossible to drill a hole to the exact size of the drill. Therefore to obtain a hole of standard size, round and smooth, it is practical to drill or bore to $\frac{1}{32}$ in. undersize and then machine ream. If greater accuracy is required, it may be bored or machine reamed to within $\frac{5}{1000}$ in. of size and then hand reamed.

Reamers are made of either carbon-tool steel or high-speed steel; in hundreds of sizes; for general or specific purposes; in various types and kinds; straight, and in all standard and many special tapers. They are made with straight or taper shanks for machine use, and with squared shank for hand reaming, and many sizes are available in either expansion or adjustable types.

Spiral teeth in a reamer cut more freely, tend to prevent chatter and to prevent catching if there is a longitudinal slot or keyway in the hole.

143. Chucking or Machine Reamers (Fig. 133).—Machine reamers are largely used in drill presses, lathes, and similar machines. There are two types of machine reamers, rose reamers and fluted reamers. In the *rose reamer*, the teeth are beveled on the end and "backed off"; they cut only on the end. The lands[1] are nearly as wide as the grooves and are not relieved (backed off). The flutes or grooves are provided for conveying oil to the cut and chips away from the cut. The rose reamer tapers slightly smaller toward the shank (about

[1] *Land.*—In reamers, milling cutters, etc., the width of the top of the tooth is called the land.

0.001 in.) to prevent binding; it does not cut a particularly smooth hole but is very useful to bring the hole to within a few thousandths of size when it may be finished with the hand reamer. Rose reamers, therefore, are usually made 0.003 to 0.005 in. under nominal size.

The *fluted reamer* has more teeth for a given diameter than the rose reamer. The lands are narrower, and are backed off

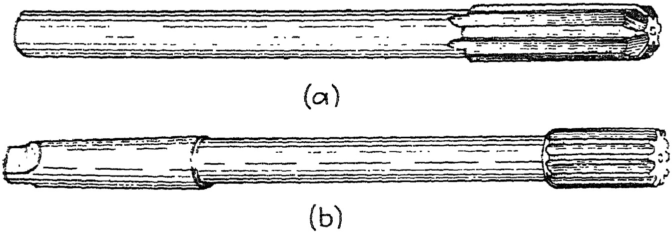


FIG. 133.—Machine reamers. (*a*) Straight-shank rose reamer; (*b*) taper-shank fluted reamer.

the whole length. The front ends of the teeth are beveled or rounded and then relieved. It is a valuable finishing reamer when extreme accuracy is not required.

Both the rose reamer and the fluted reamer are made with either straight or taper shanks. It is not usually advisable on

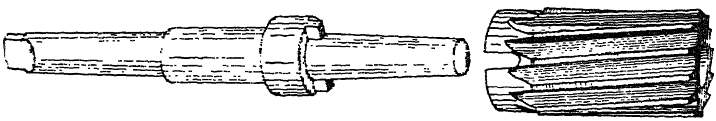


FIG. 134.—Shell-reamer arbor and shell reamer with spiral teeth.

account of the extra cost to buy taper-shank reamers under 7⁄16 in. diameter; and it is not usually good practice to buy straight-shank reamers of over 1 in. in diameter on account of the difficulty in holding them.

144. Shell Reamers.—For reasons of economy, many manufacturers prefer the shell reamers and arbors illustrated in Fig. 134. These reamers are made in either rose-reamer style or fluted-reamer style and the arbors with either straight or taper shanks, and differ in no particular respect from the

ordinary solid reamer except that one arbor may be fitted to a number of reamers and when a reamer is worn out it may be thrown away without discarding the arbor, making for economy in the end.

145. Hand Reamers (Fig. 135).—Where a particularly accurate hole is required it is first drilled or bored or machine reamed to about 0.005 in. undersize and then hand reamed.

A hand reamer is essentially a finishing tool, a scraping tool, and is ground straight for nearly the whole length of the teeth.

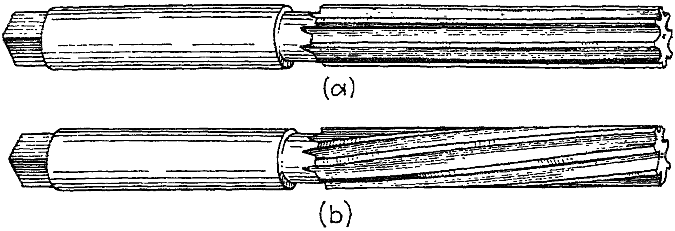


FIG. 135.—Hand reamers. (*a*) Regular; (*b*) with spiral teeth.

It is slightly tapered, smaller toward the front, for a distance about equal to its diameter, to permit of its entering the hole to be reamed. The teeth are relieved a very little for clearance. The shank end is machined square to receive the wrench. *The hand reamer should never be operated by mechanical power.* Care should be exercised to start it true and keep it straight. It is often advisable to start the hand reamer when aligned and steadied by the dead center of the lathe or a center placed in the drill-press spindle as the case may be. *Do not leave over* 0.005 *in. for a hand reamer.*

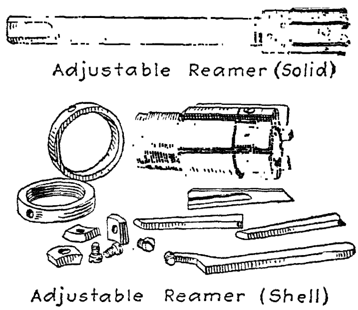


FIG. 136.

146. Adjustable Reamers (Fig. 136).—Probably the most efficient kind of reamer for any purpose is the adjustable-blade reamer. The best types of these reamers can be adjusted to

sizes within a considerable range over or under nominal size; often a valuable feature. While their first cost is considerably in excess of the solid type of reamer, the fact that they may be easily sharpened and quickly adjusted to an exact size, and their corresponding long life, make them a particularly efficient tool. These reamers are made in all standard sizes, either hand or machine, with the body and shank in one piece or of the shell-reamer variety.

147. The Expansion Reamer.—The body of the expansion reamer (Fig. 137) is bored slightly taper and slitted to permit a slight expansion (about 0.005 in. in a 1-in. hand reamer and

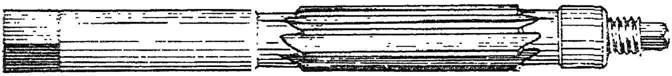


FIG. 137.—Expansion hand reamer. (*Courtesy of Morse T. D. & M. Company.*)

proportional amounts in other sizes 0.25 in. and over). A tapered plug, threaded through the end (guide) and squared for a wrench, is the expander. This reamer is not meant for

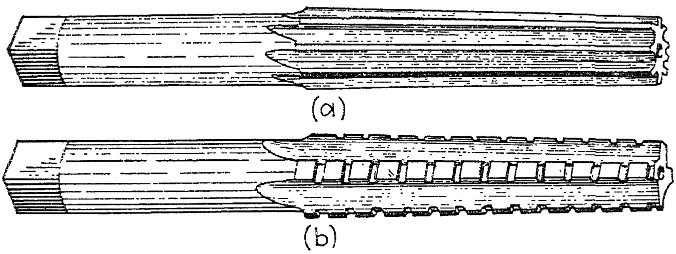


FIG. 138.—Hand reamers for taper holes. (*a*) Finishing; (*b*) roughing.

an oversize reamer, or for an adjustable reamer; it is meant to give longer life to a reamer for finishing standard-size holes.

148. Taper Reamers.—Taper reamers, for both roughing and finishing (Fig. 138), are made for all of the standard sizes of tapers. The end of the shank of the hand reamer is cut square to receive the wrench and the reamer should always be turned by hand. As the chips do not fall out readily a taper reamer should be removed often and cleaned.

By reason of the shearing cut such as is given to a reamer by the spiral flutes, there is less strain upon the reamer, less clogging with chips, and a smoother and more accurate hole results. This is especially true with steep-spiral taper reamers

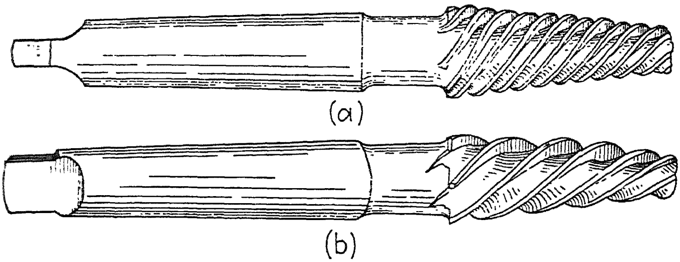


FIG. 139.—Machine reamers for taper holes. Have steep left-hand spiral teeth. (*a*) Finishing; (*b*) roughing.

for use in machines (Fig. 139). Rough with the coarser tooth and finish with the finer tooth reamer.

149. Unequal Spacing of Teeth.—Chattering is caused by more or less rapid vibration of the work or tool and usually produces a slight wave-like unevenness in the surface being finished. It is caused by improperly adjusted bearings, too much spring of the tool or work, too wide a cut, or too much clearance on the tool.

The hand reamer has a wide (long) cutting edge and several of them and consequently has a tendency to chatter. To overcome this tendency the tooth clearance is very slight. Having the teeth spirally cut seems to help also, and, further, the teeth are *increment cut*, that is, unequally spaced. With such teeth chatter marks cannot occur at the same rate or time—cannot *synchronize*, and, therefore, the tendency is for the reamer to cut smooth.

150. Drilling in a Lathe—Spotting the Center.—Suppose it is required to finish a hole, say 1 in. in diameter, in a piece of solid metal. The first operation usually, after truing the work in the chuck, is to face the piece, especially if it is cast iron, and the next operation is spotting a center for the drill, using the bent spotting tool (see Fig. 140).

This tool is ground to an angle of about 120 deg. to correspond with the angle formed by the lips of the drill. Note that each lip is given clearance but in opposite directions, because the cutting force is up on one lip and down on the other. Set the spotting tool "on center" and "square," that is, in such a position that both lips will cut evenly. Start the lathe and run the tool up to the work. If the point of the tool is not exactly in the center a small ring will be turned in the face of the work. It is very easy to adjust the point to the center of this ring. Make the spot nearly as large as the diameter of the drill to be used.

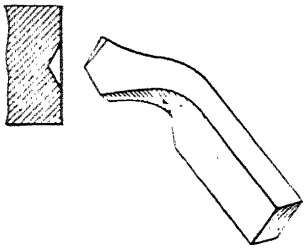


FIG. 140.—Spotting tool.

A tool bit may be ground and used for a spotting tool if it is given sufficient side clearance (Fig. 141). Care must be taken or the point, being very delicate, will break.

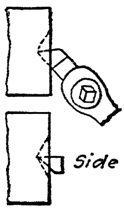


FIG. 141.

151. Operation of Drilling the Hole.—Select a drill somewhat undersize (1/64 or 1/32 in.), to allow for reaming, and a drill holder (Fig. 142) to fit the taper shank of the drill. Be sure that both the taper shank of the drill and the taper hole in the holder are clean and *free from oil.* Tapers will not hold if oily, and if the taper does not help hold the drill from turning, the tang of the drill may be twisted off under the pressure of the cut.

NOTE.—If the drill has a straight shank a dog may be used to keep it from turning, provided the drill has a good center hole. Place a protecting piece of brass or copper under the dog screw.

It is important before starting the drill to note that the tail center is not offset.

Place the point of the drill in the "spot" and the center of the drill holder on the tail center of the lathe (Fig. 143). Have the spindle well back in the tailstock, and the tailstock tightly

clamped to the bed. Upon revolving the work the drill may be fed into the work by turning the tailstock handwheel.

As the drill "breaks through" the inside end of the piece, it has a tendency to pull away from the dead center, due to the spiral and to the lack of resistance. *Do not try to hold the drill back against the center by hand because it is dangerous.* Many drills have been broken and many hands have been severely injured by ignorance or carelessness in this respect.

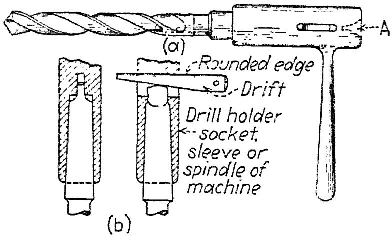


FIG. 142.—(*a*) Drill holder, fits the taper shanks of drills and machine reamers. Has a large center hole at *A* and is provided with a suitable handle. In (*b*) is illustrated how the taper shank fits in the holder, and also the use of the drift to remove the drill.

There will be no trouble if a tool clamped in the tool post is arranged against the handle of the drill holder so that the drill cannot pull away from the tailstock unless it pulls the carriage along. This setup is shown in Fig. 143. Do not wait until the drill starts to break through but arrange as above when making the setup. While the carriage moves easily enough by the screw pressure against the drill holder and does not make the feeding of the drill noticeably harder, it still offers resistance enough to keep the drill holder against the center when the drill has a tendency to dig. A further caution may here be emphasized: *Never loosen the tailstock, or withdraw the dead center from the drill while the lathe is running.* After the hole has been drilled, *stop the lathe* and keeping the drill holder against the dead center, either run the tail spindle back or loosen the tailstock and pull it back until the drill can be removed.

In lathe drilling the speed is nearly always too slow, probably due to the fact that the chuck, being so much larger than

the drill, seems to be going fast enough. Calculate the required r.p.m. and set the speed accordingly, especially until experienced.

152. Machine Reaming.—Reaming is the next operation. The machine reamer is held in a drill holder or by a dog, and should be held against any tendency to pull away from the center as explained in paragraph 151, not only when the reamer breaks through but *during the whole length of the cut.*

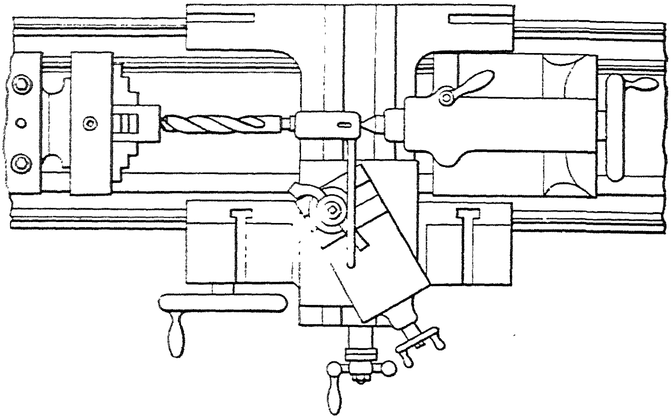


FIG. 143.—Drilling in a lathe. It is very important to have the tool post tight and the handle of the drill holder against it. Turning the tailstock handwheel will feed the drill and at the same time push the carriage along the ways.

This precaution is absolutely necessary or the reamer will catch and bend or break and the hole will be spoiled.

The speed for reaming is usually somewhat slower than for drilling especially in cast iron, to avoid any tendency to overheat and ruin the reamer. The feed should not be crowded or the reamer is likely to tear the surface of the hole.

Place the reamer in position with the end in the hole, get everything ready, then start the lathe and start to feed immediately by turning the handwheel. Ream cast iron dry, except when sometimes a little oil may be rubbed on the lands of a rose reamer to keep it from scoring. Always use a lubricant when reaming wrought metals or steel.

153. Use of Drill Chuck.—Straight-shank drills and reamers up to ½ or ⅝ in. diameter may be held in a drill chuck, the taper shank of which fits the tailstock spindle hole. If a reamer is held in this way one must be careful to have it true in the chuck or it will ream the hole slightly taper. In any case have the shank of the chuck *tight* in the spindle or it may loosen and score both shank and spindle. This setup is shown in Fig. 144.

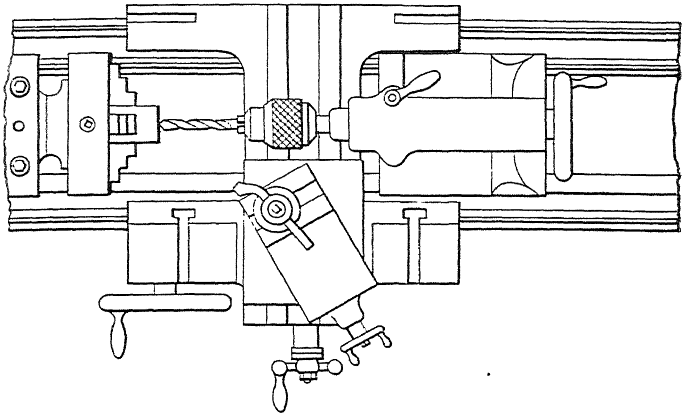


FIG. 144.—For holes under ½ in. use the drill chuck with a shank that fits the taper hole in the tailstock spindle.

154. Hand Reaming.—If for any reason it is necessary to finish the hole with a hand reamer, it is usually better to start the reamer while the work is in the lathe. Keep the dead center against the reamer and turn the reamer by hand with a wrench. After the reamer is well started the work may be removed and the hole finished in a vise.

It is especially important to *know* that the dead center is in line when starting a reamer in a lathe as above. If the dead center is even slightly offset the reamer will not start true.

Sometimes when the work is held in the vise and the reamer turned with the wrench (tap wrench, Fig. 175), a chatter will occur at the start. This may often be avoided

by holding the reamer in the vise and, with a suitable wrench or clamp, turning the work on the reamer.

A burr on the edge of a reamer may spoil the hole. When the reamer is obtained from the toolroom, feel along the edge of each tooth and if any burr is felt, oilstone it off.

Remember.—Do not leave over 0.005 *in. for a hand reamer to remove. Never turn a reamer backwards. Use a cutting lubricant when drilling or reaming steel, but cast iron is drilled and reamed dry.*

Questions on Drilling and Reaming

1. If the point of the centering tool is not exactly on center, what kind of a cut will be made in the face of the work?

2. How may the center then be easily found?

3. What shape is the end of the spotting tool? Why?

4. How is clearance on the cutting edge of the spotting tool nearer the operator ground? How is it ground on the other edge? Why?

5. How large a spot should be made?

6. If the work is a rough casting or any piece that is not fairly square, why should it be faced before drilling?

7. Can the spotting tool be used to clean a portion of the face around the spot?

8. How may a tool bit be ground to produce a satisfactory spot?

9. What is the effect in drilling if the tail center is offset?

10. How do you feed the drill?

11. Should the cutting speed be the same as if the drill revolved? Why?

12. What is the number of r.p.m. necessary to give 30 ft. per minute cutting speed for a 1-in. diameter drill? For ½-in. diameter drill? For 1¼-in. diameter drill?

13. How do you judge the proper feed?

14. As the drill breaks through at the end of the hole the tendency is for it to draw in or "dig in." This will pull the drill holder off the center and probably break the drill. How is this prevented?

15. If it is a fairly deep hole in steel, how is cutting compound applied?

16. What does a squeak indicate?

17. If a straight-shank drill is used, how may it be held with a dog? What about the center in the drill? How do you keep the drill from becoming scored by the dog?

18. How is the dog arranged on the drill? How is it arranged to keep the drill from drawing in?

BORING IN A LATHE

155. Reasons for Boring.—It is often necessary, after drilling, to enlarge a hole with a boring tool, sometimes because a drill of the proper size to leave just enough for reaming is not at hand; or because a machine reamer of the size to leave the correct amount for the hand reamer is not available; but usually for the reason of obtaining a hole which runs true.

If it is important to have a hole run true and central with the piece, as set up in the chuck or on the faceplate, the hole should be bored. It is not safe to assume that a drill will run perfectly true through solid metal, even though it starts true; it might strike a blowhole, or a hard spot in the work, or it may become dull. These conditions will cause the drill to wobble. A reamer will follow the general direction of the hole as drilled, and the result, if the hole is not straight, will be most unsatisfactory.

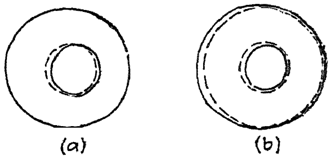


FIG. 145.—(*a*) Shows the casting trued up by the outside and the dotted line the hole (as bored) which does not clean. (*b*) Shows the eccentricity divided so that both the hole and the outside will clean.

A cored hole, if it is to be finished true, must invariably be bored. A three-lipped drill is steadier and stronger, and is therefore better than a two-lipped drill for drilling a cored hole, but the hole must be bored round and true after any drilling operation if accuracy is required.

Occasionally a core is not properly set in a mold and the hole in the casting is consequently out of center. In many such cases the casting may be adjusted in the chuck so that the eccentricity is divided, half in the hole, half on the outside, and both surfaces be finished to size (see Fig. 145). In this event the hole should be trued up by boring, at least deep enough to give a fair start for the drill, possibly its full length.

156. The Boring Tool.—The boring tool (the part that cuts) is a turning tool held in a bar or holder or forged on the end

of the bar (Fig. 146). It is ground like a turning tool that cuts from left to right. It has side rake (see *a*, Fig. 147); a cutting angle of about 60 to 70 deg. (being a metal turning

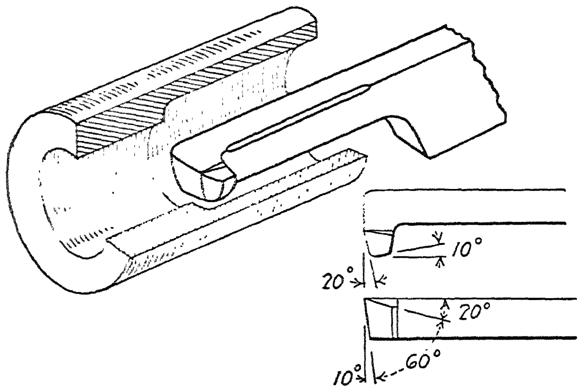


FIG. 146.—The forged boring tool.

tool), and a rounded cutting point to give the tool a longer life and the work a smoother finish. The cutting edge is not at right angles to the axis of the work but should be about 20 deg. from this perpendicular as shown in the figure. This

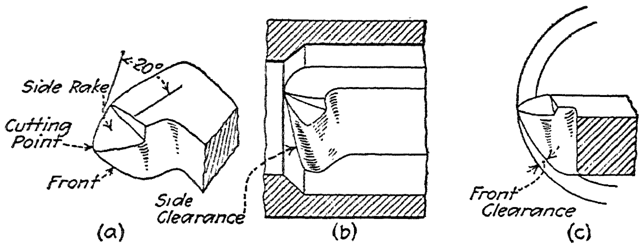


FIG. 147.

causes the chip to curl away from the finished cut and also reduces the tendency for the tool to spring into the work. The boring tool must have *side clearance* (*b*, Fig. 147) to permit of its feeding into the work and thus peel off the chip; and also *front clearance* (*c*, Fig. 147) so as not to rub on the finished work.

The clearance must be sufficient to preclude any chance of rubbing but should not be excessive or the cutting edge, not being "backed up," will break away and dull quickly.

Have the boring-tool bar as short as the length of the hole will permit and *be very sure that the bar will clear* as the tool works into the hole.

157. The Boring-tool Holder.—A boring-tool holder in which the length of the bar and the position of the cutting

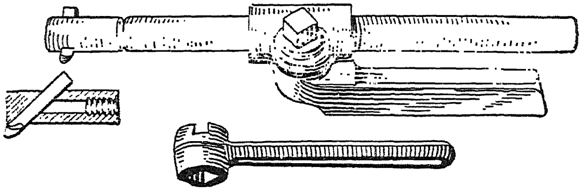


FIG. 148.—Armstrong boring-tool holder and wrench for adjusting or tightening either the cap or the bar.

edge of the tool may be adjusted will usually give better satisfaction than the forged tool. Figure 148 represents a boring-tool holder having these advantages. The tool bit may be sharpened and reset without disturbing the toolholder, which advantage is especially desirable when cutting an inside

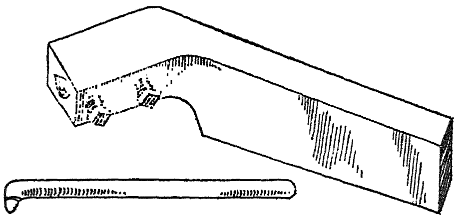


FIG. 149.

thread. The extra cap which holds the tool point at an angle is useful when boring to a shoulder or when squaring the bottom of a hole.

Figure 149 shows a "home-made" boring-tool holder which is very handy. The hole is made V shape lengthwise opposite the screws to give a perfect seat for various sizes of tools. The tools may be made from pieces of drill rod.

158. Measuring a Hole.—The size of a hole is measured with an inside caliper, and the measurement may be read on a scale as shown (*a* and *b*, Fig. 150). This is all right for the roughing cut, but when accuracy is required it is better to use a micrometer to read the size (see *c*, Fig. 150).

Do not set the caliper to size and then try to judge how much of a finishing chip to take but measure the hole and then move

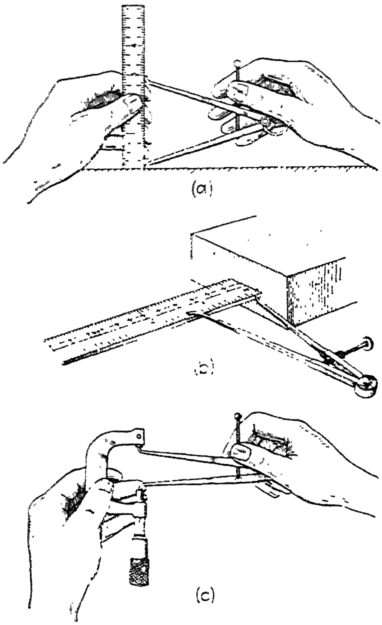


FIG. 150.—For setting or for reading the measurement of an inside caliper.

the cross slide the desired amount for the next cut. If several holes are to be bored, it is a good plan to have an extra caliper set to exact size to gauge the final cut.

Care must be exercised when measuring a hole while the work is still held in the chuck. Try to get the measurement *vertically* across the diameter. Hold the bottom leg of the caliper lightly with the forefinger of the left hand (Fig. 151).

With the right-hand thumb and forefinger adjust the caliper until the "feel" or "tickle" of the top leg is faintly felt. To get the faintest tickle, move the top leg of the caliper (wiggle it) back and forth and from side to side, fairly slow, until the caliper exactly measures the diameter. To check the measurement, hold the caliper lightly, at the end, between thumb and forefinger—this gives a more delicate touch. It is easy for a beginner to feel 0.001 in. when calipering a hole.

159. The Operation of Boring a Hole.—Select a boring tool with a shank small enough to clear the hole and not unnecessarily long. When the shank is long, the spring of the tool is excessive, and the tendency to chatter is increased.

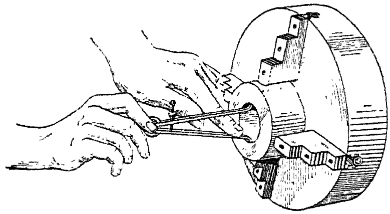


FIG. 151.—Measuring a hole with inside caliper.

Chattering, besides causing a poor appearance of the work, quickly dulls the cutting edge of the tool. It may be avoided, usually, by having a boring-tool bar of sufficient cross section, not too long, and held rigidly.

Be sure that the boring tool is sharp, and that the clearance angles and rake angles are correct for the hole to be bored. It is never safe to *assume* that a boring tool or any other tool is right; examine it and be sure it is right for the job, and particularly that it is *sharp*. The best mechanic working on the best machine cannot do efficient work with dull tools; a real mechanic will not attempt it.

Set the cutting point of the boring tool as nearly on center as possible.

The speeds and feeds for boring are substantially the same as for turning a similar material. The depth of cut, however, is usually less because of the spring of the tool.

The beginner should be warned against the tendency to bore bell-mouthed holes—holes larger at the beginning than farther along. This is caused by taking several light cuts for a short distance to obtain the correct diameter, then throwing in the feed. When the tool gets to the heavier cut, it springs away somewhat, and of course bores a smaller dimension than at the start. In any event, an extra finishing cut with a sharp tool will probably be necessary if the hole must be exact.

Care must be taken to avoid springing or breaking the work when clamping in a chuck, especially work with a thin rim or wall, and it may be advisable to ease up on the jaws before taking the finishing cut.

Questions on Boring Tools, Boring, and Reaming

1. What is the difference between drilling and boring?

2. Is the boring tool a turning tool? Where does it cut? What part of the boring tool is the front? The side?

3. Will a boring tool properly ground for a hole 2 in. in diameter have the right clearance for a hole 1 in. in diameter? Give reason.

4. If a boring tool is properly ground and set, at what angle with the axis of the work is the cutting edge? Why? Why is it on center?

5. Give two reasons for rounding the cutting point of the tool.

6. Should the tool for boring cast iron and steel be given front rake and side rake? Give reasons.

7. In what respect should a tool for boring brass differ from a tool for boring steel? Give reason.

8. In what two directions does a boring tool have a tendency to spring? How may this spring be largely overcome?

9. What particular care must be taken regarding the shank or bar of a boring tool when setting up?

10. Using a carbon-steel tool, what cutting speed is proper for boring machine steel? Cast iron? Tool steel? Brass? What cutting speed is proper for these materials if a high-speed tool is used?

11. What r.p.m. of the work is necessary when boring a hole 2 in. in diameter in cast iron? In brass?

12. How is the proper feed determined in boring? The proper chip?

13. How is the hole measured? How is the measurement read if a scale is used? If an outside micrometer is used?

14. What is a bell-mouthed hole? Why will too many trial cuts at the beginning of a hole tend to make it bell mouthed?

15. How is the graduated cross-feed screw used in boring operations?

16. What is the object of boring a hole? When is it advisable to bore a hole that is afterwards to be reamed?

17. When is a machine reamer that is up to size used in lathe work? When is a machine reamer slightly undersize used? How much undersize should it be made?

18. How is a taper-shank reamer held? How is a straight-shank reamer held?

19. Why must the reamer be held back against the center during the whole cut?

20. How does the cutting speed of a reamer compare with that of a drill? How does the feed compare?

21. When is the hand reamer used in a lathe?

22. How much undersize should a hole be left for hand reaming?

23. How is the reamer started square? What must be the exact position of the dead center? Why?

24. How is the hand reamer turned when used in a lathe? How is it kept square?

25. Why should the hand reamer not be turned backwards?

26. When should a lubricant be used on a hand reamer?

27. Why must extreme care be taken when reaming pieces which have thin walls?

28. The body of a hand reamer is not cylindrical. What part is smaller? How much smaller? Why is it smaller? Is the rest of the body cylindrical?

29. How is a hand reamer given "clearance"?

30. What is meant by a reamer with "increment" cut teeth? What advantage has it?

31. What is an adjustable reamer? What are its advantages?

32. What are the advantages of a hand-reamed hole?

33. As the chips do not fall out readily from a taper reamer what precaution must be taken?

CHAPTER IX

TAPERS AND ANGLES

160. Tapers.—One of the most important principles in machine-shop practice is that involved in taper work, particularly the round taper shank and the round taper hole (Fig. 152).

There is hardly a revolving spindle in any machine that is not provided with a taper hole. This taper hole will receive

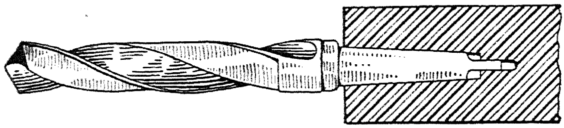


Fig. 152.

and securely and rigidly hold the taper shanks of various tools such as centers, drills, reamers, etc. The correct position of the tool thus used is immediately obtained and indefinitely maintained, yet a slight blow serves to remove the shank from the hole.

Taper in round work may be defined as the difference in diameters, for any length, measured along the axis of the work. It is usually stated in tables, on drawings, etc., as the amount of taper *per foot.*

There are four parts to every taper, the amount it tapers per foot; the length of the taper; the large diameter; and the small diameter (Fig. 153).

161. Standard Tapers.—There are various systems of standard tapers in common commercial use, the most important being the *Morse Standard,* the *Brown & Sharpe Standard,* and the *Taper-pin Standard* (see list of tables, p. 367).

Twist drills are made with Morse standard taper shanks, up to and including $\frac{9}{16}$ in. with No. 1 taper, from $\frac{9}{16}$ up to and including $\frac{29}{32}$ in. with No. 2 taper, etc.

The Brown & Sharpe standard taper is used in milling machines; the arbors, collets, end mills, etc., have shanks with Brown & Sharpe tapers, with one exception—the new standard milling-machine spindles have the hole for the arbor, etc., with a taper of $3\frac{1}{2}$ in. per foot.

There is no standard taper for the shanks of lathe centers. Each manufacturer seems to have established sizes of his own. Do not attempt to use a center made for one lathe in another kind of lathe, and do not use in a lathe a chuck with a shank fitted to a drill press. The chances are the taper will not fit.

A disturbing fact in machine work which often results in considerable confusion, is the number of mongrel sizes in the various systems. For example, there are six different amounts

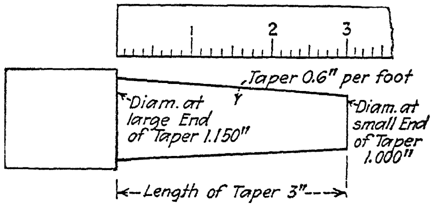


FIG. 153.

of taper per foot in the eight Morse standard tapers. No one tries to remember the lengths, diameters, etc.; a table of sizes is necessary.

The Jarno system of tapers is the most sensible system. In this series the number of the taper is the key by which all the dimensions are immediately known. Thus, the number of the taper is the number of *eighths* of an inch in diameter at the large end, the number of *tenths* of an inch in diameter at the small end, and the number of *halves* of an inch in length. The taper is 0.6 per foot in each size. It is too late to incorporate the Jarno system in drilling machines and milling machines; there are too many million drills, reamers, end mills, etc., in the shops to make a change feasible, but there seems to be no real reason why the Jarno system cannot be used in the new lathes.

To preserve the accuracy and efficiency of tapers (shanks and holes) they must be free from dirt, chips, and nicks or burrs. A most distressing sight is a taper, either a shank or a hole, practically spoiled by being nicked and dented. The most important single direction in regard to tapers is to *keep them clean*. The next important thing is to wipe them dry, because an oily taper will not hold.

TAPER TURNING

There are several methods of turning a taper in a lathe: by offsetting the tailstock slide; with a square-nose tool; by means of the compound rest; and with a taper attachment. When lathes are not provided with the taper attachment it is customary to obtain the taper by offsetting the tailstock slide. This will be the first method considered. The others will be explained presently.

162. Offsetting the Tailstock Slide.—If the dead center is offset, the center line of the work which is held on centers

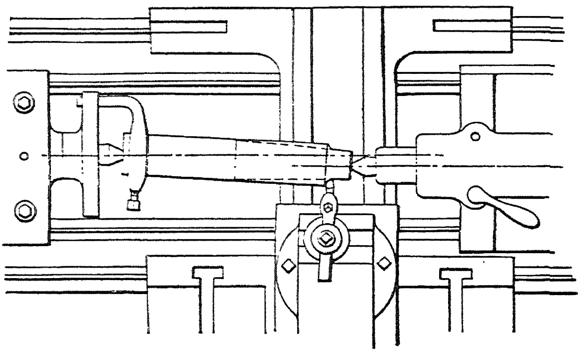


FIG. 154.—Turning a taper-tailstock offset.

will not be parallel to the line of travel of the turning tool and the work will be turned taper (see Fig. 154). To find the proper amount to offset the tailstock for a given taper on a piece of work requires a simple calculation, but there are a few things about offsetting the tailstock for turning tapers

that must be thoroughly understood before calculations can be made intelligently.

1. The more the given piece is offset the greater will be the amount of taper.

2. The longer the piece of work the more the offset required to obtain a given taper. For example: An offset of $\frac{1}{16}$ in. for a piece 2 in. long would give a fairly steep taper but the same offset for a piece 24 in. long would give a taper hardly noticeable, in fact, only one-twelfth as much as the first piece.

3. The length of the taper itself, that is the distance a taper is cut on a piece of work, has nothing to do with the offset. When making calculations do not get length of the taper confused with the length of the work.

4. Since the work revolves in the lathe only one-half as much offset is required to give the same amount of taper as if the work did not revolve.

5. The taper is proportional to the length—so much taper per foot. The offset is proportional, but in the ratio of one-half the length because the work revolves.

Measurements in machine-shop work, including lengths up to 2 ft. or more, are expressed in the denomination of *inches*. Therefore, when calculating the offset, where the length is given in inches, the taper per foot is always reduced to taper per *inch* (divide taper per foot by 12). Then the proportion will be

$$\text{Offset} : \text{taper per inch} = \frac{\text{length of work in inches}}{2} : 1 \text{ inch.}$$

that is, $$\text{Offset} : T = \frac{L}{2} : 1$$

or $$\text{Offset} = \frac{TL}{2}$$

This explains the derivation of the following:

163. Rule for Offset When Turning Taper.—Multiply the length of the work *in inches* by the taper *per inch* and divide by two. The result will be the amount to offset the tailstock.

NOTE: When measuring work and calculating offset, disregard eighths and take the nearest quarter inch under. For example: if the work measures $9\frac{7}{16}$ or $9\frac{3}{8}$ call the length $9\frac{1}{4}$ in your calculation. As a matter of fact, the centers of the lathe enter the work a little way and no doubt the offset as calculated using $9\frac{1}{4}$ will be nearer right.

164. Methods of Gauging Offset.—Assume that the lathe is set for straight turning and that a certain amount of offset is required to turn the taper. Hold the tool post rigid by clamping a tool as in Fig. 155. Run the cross slide in until a piece of paper is lightly pinched between the tool post and the tailstock spindle (*a*, Fig. 155). *Take up the lost motion in the*

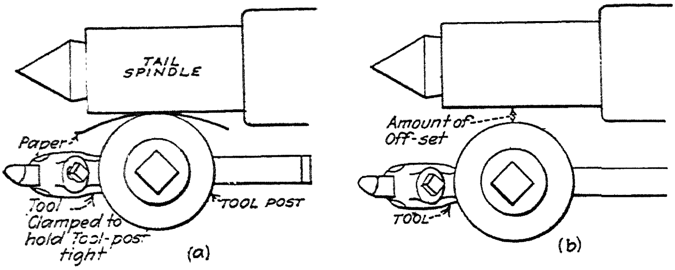


FIG. 155.

cross-feed screw, and using the graduations on the collar, run the cross slide away from the spindle until the distance between the tool post and the spindle is equal to the amount of the required offset (*b*, Fig. 155). Then adjust the *tailstock slide* until paper is pinched between the spindle and the tool post, thus obtaining the required offset.

If necessary to offset in a direction away from the operator, use a similar method. Arrange as above at *a*, Fig. 155, offset the tail spindle a little farther than necessary, run the cross slide in the proper amount, and adjust the tailstock back toward the operator until the piece of paper is pinched between the spindle and the tool post.

165. Setting of Turning Tool.—When calculating the required offset, it may be readily proved that the three lines of the problem (the center line of the lathe, the center line of the work, and the offset line) are in the same plane. There-

fore in turning or boring taper it is absolutely necessary to have the cutting point of the tool on center.

166. Methods of Measuring Tapers.—For the reason that the centers enter the work a short distance, which fact is usually ignored in calculations, and also that there is a possibility of other errors, it is necessary always to *test the amount of taper before turning the work to size.*

The taper per inch of any turned piece may be easily obtained by dividing the difference in diameters by the length in inches measured along the axis of the work between these diameters. For example: To find the taper of a sample piece, or to ascertain if a taper being turned is correct, the following method may be used to obtain an approximately accurate result. With pencil or scriber draw two lines on the surface of the taper parallel with the end, and if convenient, a whole number of inches apart. Measure the diameters at these lines and divide their difference by the number of inches between them. The result will be the taper per inch.

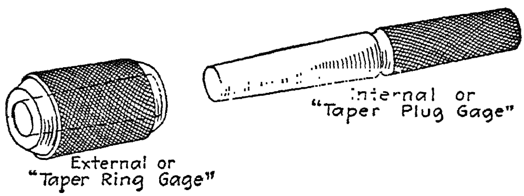


FIG. 156.

When obtaining the taper per inch it is the usual practice to consider the length as measured on the surface as near enough for practical purposes, because the difference between the length of an ordinary taper measured "along the axis of the work" and the length measured on the surface of the work is so small that it is, in most cases, not worth considering.

167. Fitting a Taper to a Gauge.—It is difficult to accurately measure the diameters of a taper with a spring caliper or a micrometer. A taper should be finally fitted to a gauge (Fig. 156), or to the spindle or sleeve for which it is intended. To

try a taper draw three light chalk lines about equidistant along the length of the work and then wring the taper (to the left and it will not stick) a part of a turn in the gauge. If the chalk marks do not rub off evenly the taper is incorrect. When extreme accuracy is required, a very thin application of Prussian-blue oil paint may be used instead of the chalk marks.

Standard taper gauges, external and internal (Fig. 156), are practically indispensable where accurate taper work is done.

168. Gauging the Size of a Taper.—A very quick, accurate method of gauging the *size* of a taper is to note the distance it goes into the gauge. If too large, it will not go in far enough; if too small it will go in too far. For example: say a shank 0.6 in. per foot taper (0.050 per inch) does not enter the gauge within 0.5 in. of correct *depth*, it is 0.025 in. too large in *diameter*; if it goes into the gauge 0.2 in. too far it is 0.010 in. too small in diameter. Be sure to understand this. For instance, how much too large in diameter is a tapered piece that sticks out of the gauge one inch too far?

169. Duplicating a Taper Piece.—When a taper on a piece of work is to be duplicated, if it has centers it may be put in the lathe and the offset of tail spindle, or the adjustment of the taper attachment, or of the compound rest, be quickly obtained by means of an indicator placed in the tool post. When the setting is correct, the reading of the indicator will not change when moved along the length of the taper.

170. Turning a Taper with a Square-nose Tool.—It often happens that the easiest and quickest way to get an abrupt taper or angular cut on a piece is by means of a square-nose tool. For example: The live center may be trued up with a square-nose tool very efficiently (this is illustrated in Fig. 85, page 130). It is only necessary to have a fairly broad square-nose tool properly sharpened, set on center and to the desired angle.

171. Filing a Taper.—Most round work is either turned or ground to size. There are times, however, when a few strokes with a file will serve to fit a taper, that is nearly right and

wanted in a hurry, much more quickly than it could be turned or ground.

THE TAPER ATTACHMENT

172. Introduction.—The taper attachment (Fig. 157) has many features of especial value among which are the following: (1) The lathe centers are in line and the center holes in the work are not distorted. (2) The length of the work need not be considered, for once the taper is set, that particular taper

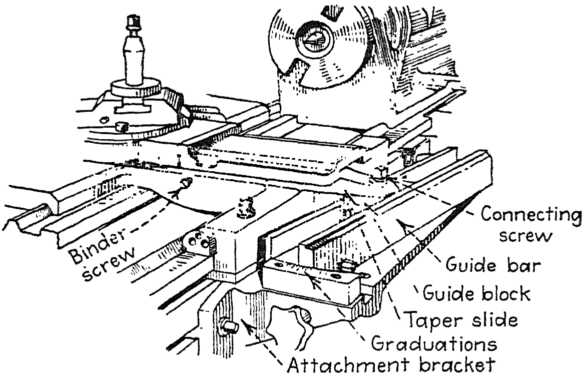


FIG. 157.—Taper attachment. The taper slide acts also as the connection, see Fig. 160.

will be turned on any length of piece. (3) The alignment of the lathe need not be disturbed, thus saving considerable time and trouble. (4) Taper boring is accomplished as easily as turning. (5) A much wider range is possible than by the offset method; for example, to turn a taper ¾ in. per foot on the end of a bar 4 ft. long would require a setover of 1½ in. which is outside the limit of a regular 14- or 16-in. lathe. Further, it is often convenient to use a combination of the offset of the tailstock and taper attachment when turning tapers too steep for either method alone.

Ordinarily, when the lathe centers are in line, the work is turned straight, because as the carriage feeds along, the tool is always the same distance from the center line. The purpose of the taper attachment is to make it possible to keep the lathe

centers in line, but by freeing the cross slide and then guiding it (and the tool) gradually away from the center line, cause a taper to be turned as in *a*, Fig. 158, or guiding it gradually nearer the center line, as in *b*, cause a taper hole to be bored.

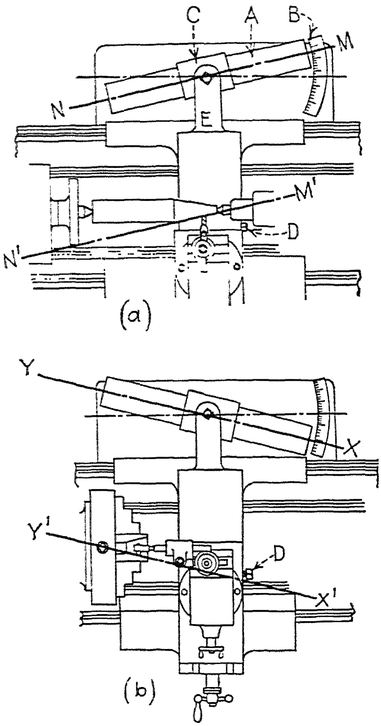


FIG. 158.—Using the taper attachment; (*a*) turning a taper; (*b*) boring a taper hole.

By *freeing* the cross slide is meant the loosening of binder screws, the *position* of which, and the *number* (in some lathes one, in others half a dozen), depend upon the design of the attachment. The purpose of the binder screws is (1) to *bind* the cross slide so it may be moved only by turning the cross-feed handle, or (2) when loosened, to *free* the cross slide for use with the taper attachment. When the cross slide is free

it may be moved an inch or more by pushing against the tool post.

173. Parts of the Taper Attachment.—There must be a guide to control the movement of the cross slide, and there must be a connection of some kind between the guide and the cross slide. It is the combination of the free cross slide, the guiding arrangement, and the connection, that makes the taper attachment. There are a number of designs of these attachments but the principle of operation is the same for all. The main features as shown in Fig. 158 are as follows:

1. The *guide-bar unit*, *A*. The guide bar is pivoted at its center; it may be swiveled to the desired position and rigidly secured. In some lathes the bracket upon which the guide bar rests is bolted to a planed surface on the back of the bed, and in others is carried on the back of the carriage. In the latter arrangement the guide bar, when in use, is anchored through a rod and locking arm to the rear V on the lathe bed.

2. *Graduations*, *B*, at one or both ends of the guide bar indicate the setting in taper per *foot* (not per inch). Usually graduations in degrees are also provided.

3. The *guide block* (or sliding shoe), *C*, is fitted to slide along the bar. (The guide *block* fits in a groove in the guide bar and a *shoe* fits over the sides and top of the bar.)

4. The *cross-slide unit* (sectional view shown in Fig. 160) is the regular cross slide with a few additional parts and so designed for the taper attachment that it may be loosened and is then quite free to slide (without turning the cross-feed screw).

5. *A binder screw* (or screws), *D*, for loosening, or tightening, the cross-slide unit.

6. A *connection*, *E*, between the cross-slide unit and the guide block.

174. Types of Connections.—If the student understands that the cross slide must be free and then is connected to the block or shoe on the guide bar, he can read the following and note which of the three types of connection is used in his lathe.

1. A *yoke* connection between the regular cross slide and the guide. The disadvantage is that the cross feed cannot be used for additional depths of cuts. In most cases, however, the compound-rest feed may be used.

2. The use of the *"telescopic" feed screw* in which the cross-feed screw is connected to the guide (see Fig. 159). By this arrangement it is possible to use the regular cross feed for additional cuts, but for heavy work the pull of the connection puts a considerable strain on the screw.

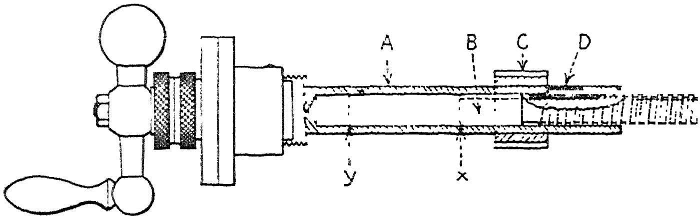


Fig. 159.—Telescopic feed screw, the screw B telescoping into the sleeve A. The feed screw is extended to the connection with the guide block and thus governs the movement of the cross slide for the given taper. The lines x and y indicate different position of the screw for the larger and smaller diameters of the taper. In any position the cross slide may be fed in or out as usual because the key D feathered in the screw turns the screw and moves the cross slide.

3. The use of an extra sliding member, called the *taper slide*, between the cross slide and the saddle (see Fig. 160). This is the oldest and simplest type, perhaps the most costly, but it has neither of the objections of the others.

175. Using the Taper Attachment.—Do not cut the taper to size until the *fit* has been checked by gauge or otherwise. The graduations at the end of the bar are for convenience, not for a high degree of accuracy.

Just as much judgment and care must be exercised in fitting a taper when using the taper attachment as when cutting a taper by any other method.

To set up and use the taper attachment, proceed about as follows:

1. The guide block should not project over the end of the guide bar either at the beginning or at the end of the cut, so make sure its position is right.

2. Clean and oil the guide bar and block.
3. Set the bar for a trial cut, using the graduations.
4. Loosen the binder screws (and remember when loosening a screw that one turn is as good as half a dozen turns).
5. Oil the flat bearing surfaces of the cross-slide unit.
6. Fasten the connection to the guide block.
7. Adjust the work between centers, and set the tool.
8. Take up the lost motion and proceed to take the first cut.

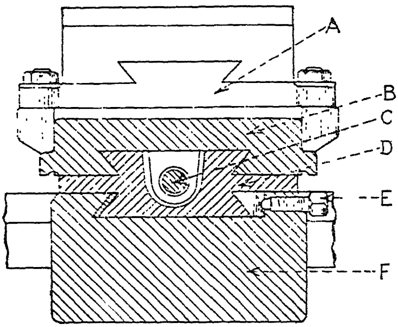


FIG. 160.—The taper slide *D* is provided between the cross slide *B* and the saddle *F*. When the taper attachment is not being used, *D* is tightened by the binder screw *E*, and the cross slide *B*, when fed either in or out, moves on *D* as a base. When the binder screw is loosened, the unit comprising *A*, *B*, *C*, and *D* is free to slide on the saddle *F* as a base. In this design the taper slide *D* serves also as a *connection* (see Fig. 157), and when fastened to the guide block it is no longer free but is controlled by the movement of the guide. The cross-feed screw *C* serves equally well whether or not the taper attachment is being used.

176. Taking Up Lost Motion.—When using certain kinds of taper attachments, the machinist's enemy "lost motion" or "backlash" must be taken care of, or serious trouble will result. In every slide and every freely revolving screw there is a certain amount of lost motion. This is very noticeable if the parts are worn. In a taper attachment so designed that lost motion may occur anywhere between the tool and the guide, care must be taken that the lost motion is taken up in the right direction before proceeding to cut. *Otherwise the piece will be turned or bored straight for a short distance before*

the taper attachment begins to work. To take up lost motion when turning taper, run the carriage back toward the dead center at least $\frac{3}{4}$ in. (use a half center if the diameter of the work is small), then feed forward by hand until the beginning of the cut, when the power feed may be thrown in. This operation must be repeated for every cut.

177. Boring Tapers with Taper Attachment.—The best way to bore a taper in a lathe is by the use of the taper attachment (*b*, Fig. 158). Extreme care must be exercised that the backlash or lost motion is taken care of when tapers are being bored with the taper attachment. Otherwise the hole will be bored straight for a certain distance, before the taper starts. Be sure that the boring tool is small enough to operate without rubbing at the small end of the hole.

178. Boring Tapers with Compound Rest.—Another method of boring a taper and one often used for very abrupt tapers (or angles) is by means of the compound rest. The compound rest is set around a certain amount to cut the desired taper or angle (see page 210), and the boring tool is fed by hand.

179. Fitting Taper Holes.—Taper holes are fitted to *taper-plug gauges* similarly as taper shanks are fitted to taper-ring gauges (see paragraph 199).

Taper holes are usually finished by reaming. For description of taper reamers, see page 180.

Questions on Tapers

1. Having found the taper per inch, how do you know the taper per foot?

2. How many parts, that is, dimensions, must be measured to determine the amount of taper?

3. When measuring a tapered piece with a caliper or micrometer, what care must be taken?

4. In measuring the taper why do you lay off a whole number of inches and not such a distance as $3\frac{9}{16}$ in., or $4\frac{3}{32}$ in.?

5. What is the taper per inch of 0.6 in. taper per foot? Of $\frac{1}{2}$ in. taper per foot? Of $\frac{3}{4}$ in. taper per foot?

6. What is the taper per foot of 0.050 in. taper per inch? Of 0.042 in. taper per inch? Of 0.062 in. taper per inch?

7. How do you try a taper in a gauge? Why do you make three chalk marks? Why not one chalk mark? Why not cover the taper with chalk?

8. If the taper fits, but is too large and does not go in the gauge far enough, how do you determine from the amount it sticks out how much more to turn off?

9. Suppose the shank of a reamer is 0.6 in. per foot taper, and that it is required to leave 0.010 in for grinding, how much farther into the gauge will the shank go after it is ground?

10. Is there a taper hole in the spindle of the milling machine? Drilling machine? Grinding machine? Is there any revolving spindle in the shop which does not have a taper hole?

11. What are some of the cutting tools held by means of tapers? Are they held securely? Is the friction of the taper alone sufficient to hold them?

12. What are some of the advantages of the taper in machine-shop work?

13. What is one of the most important considerations regarding tapers? What does a mechanic think of an ill-fitting or a damaged taper? What does he think of a taper that is nicked or burred?

14. Why are tenons or "tangs" milled on the ends of twist drills, machine reamers, and end mills? Why not on lathe centers?

15. What do you understand by the term "standard taper"? Name two systems of standard tapers in commercial use for holding cutting tools and state their chief difference.

16. Why will the Brown & Sharpe taper not fit in a drill press? Why will the taper shank of a chuck fitted to a lathe spindle not fit in a milling-machine collet or in a drill socket?

17. What common cutting tools are provided with Morse standard tapers?

18. In what machines are Brown & Sharpe tapers mostly used?

19. Do lathe centers have any standard taper?

Questions on Taper Turning

1. If the dead center is in line with the live center, a cylinder is turned. Why?

2. If the dead center is offset, what shape is turned? Why? What does the position of the dead center determine?

3. If the center is offset toward the operator which end of the taper is smaller? If the offset is in a direction away from the operator which end is smaller?

4. When offsetting the tailstock, how do you take care of the lost motion in the cross-feed screw?

5. If a piece of work is 1 ft. long and the dead center is offset ¼ in., the taper turned on this piece will be ½ in. per foot. Why?

6. If a piece of work is 2 ft. long and the offset of the tailstock is $\frac{1}{4}$ in., what will be the taper per foot? Why?

7. In the two preceding questions with the same offset we have two different tapers. Give reason.

8. Suppose the machinist has two pieces of steel, one 12 in. long and the other 24 in. long, and it is required to turn the same taper on each piece. What will be the difference in the offset? Which piece will require the more offset? Why?

9. If the dead center is offset $\frac{1}{4}$ in. and a piece of work is turned taper a distance of 4 in., and another piece, of the same size and the same offset, is turned taper a distance of 6 in., will the taper per foot be the same in both cases? Give reason.

10. It may be stated that two factors, one of them being the offset, determine the amount of taper that will be turned. What is the other factor?

11. Does the length of the taper have anything to do with the offset? Give reason.

12. What is the rule for offsetting the tailstock when turning taper?

13. In the above rule, why divide by two?

14. How is the amount of taper given on drawings? How is it expressed in charts of standard tapers?

15. A piece of work $7\frac{1}{2}$ in. long is to have a taper 4 in. long $\frac{1}{2}$ in. per foot. What offset is required?

16. Calculate the amount of offset for the following:

a. Length of work $8\frac{1}{2}$ in.; taper 0.6 in. per foot.

b. Length of work $6\frac{1}{2}$ in.; taper $\frac{1}{2}$ in. per foot.

c. Length of work 9 in.; taper $\frac{3}{4}$ in. per foot.

d. Length of work $6\frac{3}{4}$ in.; taper 0.5 in. per foot.

17. Two pieces of the same length are to be turned taper; one has large centers, the other has small centers. Which will require the more offset? Why?

18. If calculations are exactly right and setover of tailstock is made accordingly, and to a thousandth of an inch, is it safe to assume that the taper will be correct and therefore turn to size before trying it? Give reason.

19. To make sure the correct taper is being turned, how may the piece be measured with a caliper or a micrometer?

20. If a micrometer is used, why is the measurement made with the edges of the spindle and anvil?

21. Why should the cutting tool be set on center when turning taper?

22. If a tapered piece is to be duplicated, how may the tailstock be adjusted without calculating the offset? Can this be done if the new piece is longer or shorter than the sample? Give reason.

23. Under what circumstances only is it proper to fit a taper by filing?

24. If the centers of the lathe were in line, but as the tool was fed along it worked back gradually and uniformly, what shape piece would be turned?

25. In the taper attachment, how is the block guided? How is the guide bar pivoted? How adjusted? How is it tightened? Why is it tightened?

26. Explain in detail the principle and construction of the taper attachment.

27. State at least four advantages of the taper attachment.

28. On what sort of a taper would the use of both the offset and the taper attachment be advisable?

TURNING ANGLES

180. Angles.—An angle is the amount of the divergence between two straight lines that either meet in a common point, or would meet if sufficiently prolonged. The straight lines are called the sides of the angle and the meeting point is called the vertex of the angle. An angle is *measured* on the circumference of a circle drawn with the vertex as a center. The sides of the angle lay off a certain portion of the circumference. The circumference is divided into 360 parts or degrees

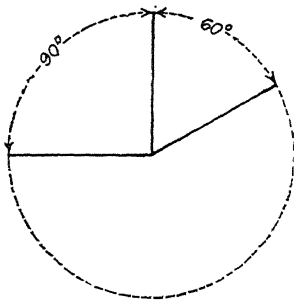


Fig. 161.

and the number of degrees between the sides of the angle is the measure of the angle. For example, if one-fourth of the circumference is intercepted between the sides, the angle is measured by 90 deg. (one-fourth of 360 deg.), and is commonly spoken of as an angle of 90 deg., or a right angle (see Fig. 161). Also if one-sixth of the circumference is intercepted by the sides of the angle, the angle is measured by 60 deg., or in other words, it is an angle of 60 deg. (Fig. 161). For fine angular measurements, the degree is subdivided into 60 parts called *minutes* and the minutes are subdivided into 60 parts, called *seconds*. The notations used are degrees (°), minutes (′), seconds (″). The subdivisions of seconds are not used in ordinary machine work.

181. Classification of Angles.

A *right angle* is equal to an angle of 90 deg.

An *acute angle* is less than 90 deg.

An *obtuse angle* is greater than 90 deg.

Two angles are called *complementary angles* when their sum is equal to a right angle and each is called the *complement* of the other. For example, 55 deg. is the complement of 35 deg.; 35 deg. is the complement of 55 deg.; 40 deg. is the complement of 50 deg., etc.

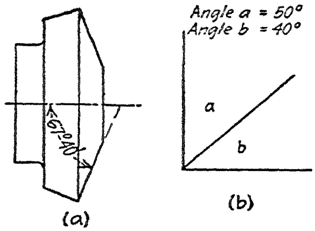


FIG. 162.

Two angles are called *supplementary angles* when their sum is equal to two right angles (180 deg.). For example 55 deg. is the supplement of 125 deg.

On drawings and blueprints, the angle is usually dimensioned as shown in *a*, Fig. 162. In formulas and calculations, it may be named by a small italic letter as shown in *b*, Fig. 162.

Tapers and Angles.[1]—Tapering pieces up to an included angle of 8 deg.,[2] which is about 1¾ in. taper per foot, are known as tapers and are measured as having "taper per foot." Pieces which are turned or bored to an included angle of greater than 8 deg. are usually spoken of and are measured as having *included angle* or as having an *angle with the center line* (the angle with the center line is half the included angle).

To illustrate: On almost all lathe centers, the shank is turned *taper* 0.6 in. per foot, and the center is turned to an *angle* of 60 deg. (60 deg. is the included angle; the angle with the center line is 30 deg.).

182. The Use of the Compound Rest for Turning Angles.—The best method of turning an angle on a piece of work is to use a tool rest that may be swiveled on the cross slide to any desired angle. This tool rest is known as the *compound rest*

[1] For table of tapers and corresponding angles see page 391.

[2] A tapered piece over 8 deg. included angle approximately will not hold in a taper hole.

(Fig. 163). The swivel plate of the rest is graduated in degrees, and the zero mark, which is usually on the side of the plate, is in line with a mark on the cross slide when the compound rest is at right angles to the center line of the lathe. There are a number of methods, used by various lathe manufacturers, for graduating the compound rest, but it does not make much difference whether the zero is on the front or on either side so long as the zero on the degree graduations coincides with the zero on the slide when the slide is set at right angles to the center line of the lathe. However, it is necessary to remember, when sitting for a given angle, that there may be any one of several *numbers*, due to the several methods of graduating, that may represent the correct setting. Graduating the compound rest is a feature of the lathe that needs standardization.

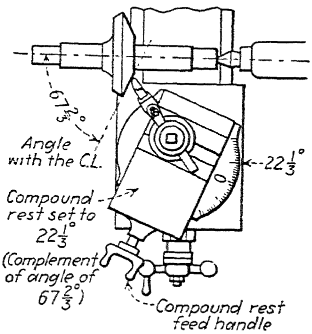


FIG. 163.

183. Setting the Compound Rest.—Setting the compound rest is comparatively easy if the operator realizes, first, that the compound rest is normally set at 90 deg. from the center line of the lathe, and, second, that the travel of the tool is to be at a certain angle other than 90 deg. with the center line of the lathe. In order to cut at a certain angle with the center line of the lathe the compound rest must be swiveled either (1) the number of degrees which is the *complement* of the angle with the center line, or (2) 90 deg. plus *the angle* with the center line.

To illustrate (1): Suppose that a bevel gear is to be turned to an angle of 67 deg. 40 min. with its axis, that is 67 deg. 40 min. ($67\frac{2}{3}$ deg.) with the center line. The complement of $67\frac{2}{3}$ deg. is $22\frac{1}{3}$ deg. Set the compound rest around $22\frac{1}{3}$ deg. from its normal position (see Fig. 163).

To illustrate (2): Suppose it is required to turn a lathe center 60 deg. included angle, the angle with the center line

is then 30 deg. Swivel the rest, to the right, 90 deg. from its original position, then swivel it 30 deg. more as shown (Fig. 164). If the compound rest were swiveled 60 deg., the complement of 30 deg., to the right it would be necessary to run the lathe backward and turn on the back side of the center. This is sometimes done. If the compound rest were set 60 deg. to the left, the handle would probably interfere with the faceplate.

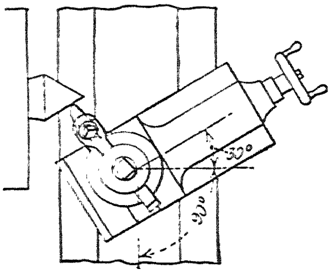


Fig. 164.

There are different methods of dimensioning the degrees of the angles on a drawing. The dimension may be given as an included angle, *a*, Fig. 165; as the angle with the axis, *b*, Fig. 165; or as the angle with a line perpendicular to the axis, *c*, Fig. 165. It is always best to find *the angle with the center line* and then set the compound rest in one of the ways

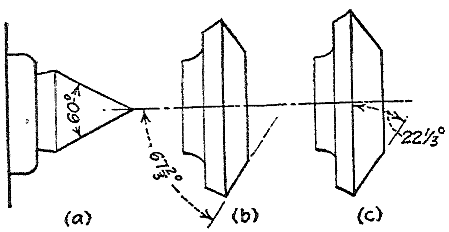


Fig. 165.

suggested in the preceding illustrations. A sketch will help in determining the correct position at which to set the compound rest.

184. Turning the Angle.—The compound rest is provided with a hand feed which is independent of the cross feed. Turning an angle is accomplished by turning the compound-rest handle. This is hand feed; there is no power feed for the compound rest. The tool will feed at the angle for which

the compound rest is set, but remember, *not always according to the figures on the graduations.*

If fairly heavy cuts are to be taken, it will be a good idea to tighten the carriage clamp screw. After the first cut is

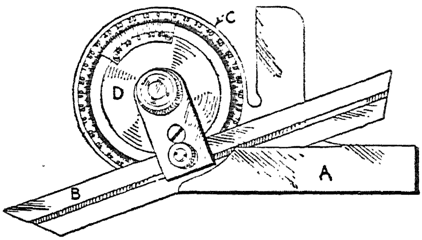


FIG. 166.—Brown & Sharpe bevel protractor. The vernier reading of this protractor is explained in the Appendix, page 361.

made run the tool back to the starting point by turning the compound-rest handle, then feed in to take the next cut by moving the cross-feed handle. Be careful not to take too deep a cut, especially near the finish.

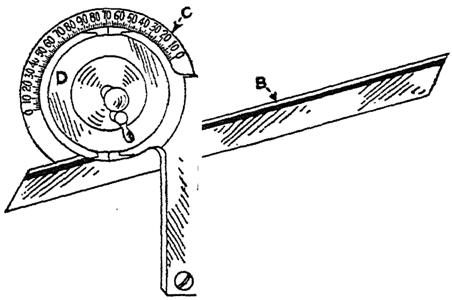


FIG. 167.—Starrett bevel protractor.

185. The Bevel Protractor.—The instrument used in machine shops for measuring angles is called the bevel protractor (Figs. 166 and 167).

The principle of construction of a bevel protractor is as follows: Two members, which may be called the beam *A* and the blade *B* with edges straight and parallel, are so arranged as to swivel on a pivot at the center of the dial *C*

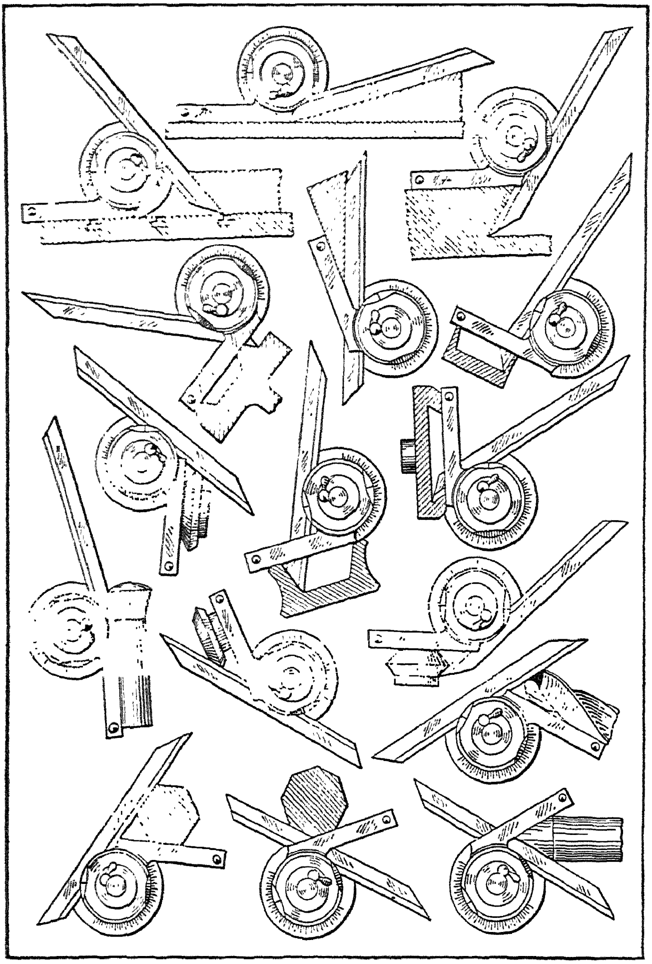


FIG. 168.—Illustrates a variety of uses of the bevel protractor.
The beginner is cautioned to use particular care when reading the angle, especially around 45 deg. For example, if the angle measures 43 deg. do not read it as 47 deg.
One side of the protractor being flat makes it convenient for laying flat on the work when scribing layout lines, or on the paper when drafting. (*Courtesy of The L. S. Starrett Company.*)

which is graduated in degrees. When the edges of the beam and blade are parallel, a small line on the swivel plate *D* coincides with the zero line on the dial, and when any measurement of an angle between the beam and the blade *of* 90 *deg. or under* is desired, the reading may be obtained direct from the position of the line on the swivel plate with regard to the graduation numbers on the dial. *But remember this:* To obtain the measurement of the angle between the beam and the blade of *over* 90 deg. subtract the number of degrees as indicated on the dial from 180 deg. This is because, as will be noted, the dial is graduated from opposite zero marks to 90 deg. each way.

Questions on Angles

1. What is an angle and how is it measured?

2. What is an acute angle? An obtuse angle?

3. In what way is an angle usually dimensioned on a drawing?

4. What is meant by the term "included angle"? "Angle with the center line"?

5. What is meant by the complement of an angle? Complementary angles? Give an example of each.

6. How is the compound rest of a lathe graduated?

7. If the compound rest is set around 30 deg. from normal position, what will be the included angle of the piece turned?

8. Why must extreme care be taken when setting a compound rest to turn a given angle?

9. Why is it best before setting the compound rest to determine at what angle with the center line the given cut is to be?

10. Explain the principle of the construction of the bevel protractor.

11. What caution must be taken when reading the measurement of an angle over 90 deg. on the bevel protractor?

CHAPTER X

THREADS AND THREAD CUTTING

Thread work is a most important part of machine-shop practice. This chapter includes: (1) Terminology, the definitions and symbols of the parts of a thread, and the terms used in connection with making threaded parts; (2) a discussion of shapes and sizes; (3) dimensions and calculations for the parts of a thread; (4) taps and dies for cutting threads; (5) gearing a lathe for cutting threads; (6) cutting a thread in a lathe; (7) measuring a thread; (8) brief discussions of various threading operations.

All of these are related thread subjects which an intelligent machinist must understand. It is not one lesson but is at least a dozen lessons. It is not difficult to learn about threads and thread cutting but it does call for real understanding. It cannot be learned by reading the chapter through once or twice, it must be studied.

186. Threads.—A *screw* is a cylindrical bar into which a helical groove is cut or formed leaving a ridge commonly called a thread. In all screws the profile of the groove is substantially the same as the thread.

A *nut* or any inside-threaded piece is the reverse of the screw, having the thread formed on the hollow inside or hole, and fitting, more or less snugly, thread to groove over the screw of corresponding size.

The screw together with an inside-threaded piece, may be used for obtaining and maintaining pressure; or to transmit motion; or as a fastening agent.

A thread on a screw may be formed by rolling in a special machine; by cutting with a revolving cutter as in a thread-milling machine; by turning in a lathe; or by means of a master

tool called a die.[1] In machine-shop practice the two methods last named are in common use.

Threads may be formed on the hollow inside as in a nut by boring in a lathe, but the usual practice is to form the threads by means of a master tool called a tap.[2]

There are various common forms of threads used for different purposes and in different localities; for example, the lead screw of the lathe is probably an "Acme" thread; the cross-feed screw is usually a "square" thread; the thread on a pipe is a modified "V" thread; the threads on most of the bolts, screws, etc., made in the United States are "American Standard," and the commonly used thread in England is the "Whitworth Standard." These threads will be explained presently.

187. Thread Standards.—A generation or so ago the common bolts and screws were cut with a thread tool that was ground theoretically to a sharp V with an angle of 60 deg. and were cut deep enough to bring the thread to a more or less sharp V. Finding it impossible to keep or even make thread-cutting tools, taps, and dies with even a fairly sharp V point, the manufacturers adopted various modifications of the theoretical V form. They rounded or flatted the point of the tool, and cut the thread on the screw and on the tap a trifle less deep thereby leaving a small flat on top and bottom of the thread.

Since each manufacturer made screws or taps or dies to suit his own purposes, the screws made in one place might not fit taped holes made in another shop or perhaps even in the same shop.

Further, there was no standard pitch;[3] one man might like the looks of 10 threads per inch on a ¾-in. bolt, another man might prefer 9, and another 12, etc. Therefore, if a bolt or screw in a machine was broken, one to replace it could not be obtained from stock or bought in the open market, it had to be specially made and fitted.

[1] *Threading Die.*—For description see page 234.

[2] *Tap.*—For description see page 230.

[3] *Pitch of Thread.*—The distance from a point on a thread to the corresponding point on the next thread.

As there was, then, no uniform modification of the 60-deg. V thread and no standard pitch for each diameter, interchangeability was practically unknown until about 1869. At this time William Sellers of Philadelphia suggested a uniform flat on top and bottom equal to one-eighth of the *pitch* of the thread, and John Fritz of Bethlehem, Pennsylvania, who was having some machines made in the Sellers Shops, suggested a standard pitch for each diameter 1/4 in. and over, for example, 1/4–20; 5/16–18; 3/8–16, etc. These ideas met with the approval finally of most engineers and have been adopted by the army and navy departments, by the railway systems, and by the manufacturers. This shape of thread and size of thread for the given diameter were for many years known as the U. S. Std. *form* of thread and the U. S. Std. *pitch* respectively.

The automobile manufacturers in 1911 adopted a system, called Society Automotive Engineers Standard (S. A. E. Std.) of finer pitches than U. S. Std. The American Society of Mechanical Engineers adopted in 1907 a system (A. S. M. E. screw threads) of standard pitches for screws under 1/4 in. in diameter. Both systems used the U. S. Std. *form* of thread.

These systems have all been brought together as the *American Standard Screw Threads*—National Coarse and National Fine. The change from the term "U. S. Standard" for pitch, and for form of thread, to the present use of the terms "American" and "National" came about through studies made by engineers over a period of years, to establish standard allowances and tolerances[1] for various classes of threads. New standards were set up and incidentally a number of changes were made in the terminology.

Briefly, the report of the National Screw Thread Commission (1921) recommended, among other changes, the substitution of the terms *National Coarse* for the "U. S. Std." screw threads 1/4 in. and over, and also for the coarser pitches of machine screw threads such as 8–32 and 10–24; and *National Fine* for the "S. A. E. Std." and the finer pitches of machine screws

[1] See definitions 21 and 22, pages 223 and 224.

such as 8–36 and 10–32—mostly those known since 1907 as the "A.S.M.E." system. They also recommended the term *National Form* for the existing 60-deg. angle of thread with the flatted top (crest) and bottom (root).

The Sectional Committee on the Standardization and Unification of Screw Threads, sponsored by two great engineering societies, carried on the work started by the N. S. T. C and after extensive surveys made a report. The American Standards Association has approved and printed the standardizations set up; and the definitions of terms, the notations used, the tables of sizes, allowances, tolerances, etc., have been adopted and published in catalogues, handbooks, and texts as for *American Standard Screw Threads*. The terms *National Coarse* (NC), *National Fine* (NF), and the *American National Form* (N) are retained.

The terms U. S. Std., S. A. E. Std., and A. S. M. E. Std. are now obsolete. Other changes may be noted in the definitions and notations (see pages 221 and 226).

To meet the requirements of fine pitches on large diameters there have been standards established for 8-pitch- (diameters 1 to 6 in.), for 12-pitch- (diameters ½ to 6 in.), and for 16-pitch- (diameters ½ to 4 in.).

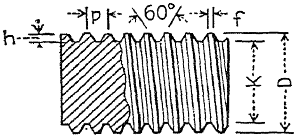
The 8-pitch thread is used for fastenings against pressure, such as cylinder-head studs and high-pressure pipe flanges.

The 12-pitch thread is used in boiler work for retapping worn stud holes 1/16 in. larger. It is also used in machine shops for thin nuts on shafts and sleeves.

The 16-pitch thread is used for fine threads needed for adjusting collars and for bearing retaining nuts.

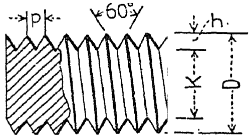
These threads, with the diameter increment in each series averaging ⅛ in., should meet the great majority of requirements and should be adhered to wherever practicable.

The U. S. Std. form, now the American National form, of thread has been adopted as the standard form of the International System and the French System. These systems are metric threads and the pitches are therefore slightly different from American Standard screw-thread pitches.



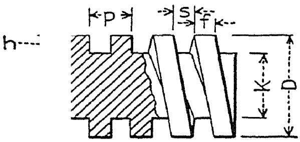
Am. (National) Standard Thread

n = Number of threads per inch
p = Pitch = $\frac{1}{n}$
h = Depth = 0.6495 × p.
f = Width of basic crest and root
D = Major diameter
E = Pitch diameter = D minus h
K = Minor dia = D − 2h = D − 1.299 × p



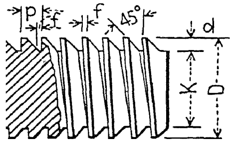
Sharp V (Theoretical) Thread

n = Number of threads per inch
p = Pitch = $\frac{1}{n}$
h = Depth = 0.866 × p
D = Major. diameter
E = Pitch diameter = D minus h
K = Minor dia. = D − 2h = D − 1.732 × p



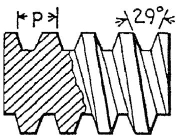
Square Thread

n = Number of threads per inch
p = Pitch = $\frac{1}{n}$
h = Depth = ½ p
f = Width of thread = ½ p
s = Width of space = ½ p
In screw S is slightly wider
In tap f is slightly wider
K = Minor diameter = D minus p

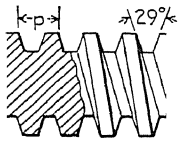


Buttress Thread

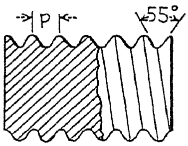
n = Number of threads per inch
p = Pitch = $\frac{1}{n}$
h = Depth = ¾ p
f = Flat (crest and root) = ⅛ p
K = Minor diameter = D − 1½ p



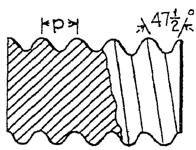
Acme 29° Thread
See Tables 10 and 11



Worm Thread B&S 29°
See Table 12



British Standard Whitworth Thread
See Table 14



British Association Screw Thread
See Table 15

For further information consult index, and also tables 7 to 17 inc. in back pages of this book

FIG. 169.

In England the standard thread is known as the *Whitworth Standard*. It was devised in 1841 by Sir Joseph Whitworth, undoubtedly the foremost mechanic of his time. It is quite unlike the American National form, having an angle of 55 deg. instead of 60 deg. and having a round top and bottom (crest and root) instead of the flat as in the American National form.

The chart (Fig. 169) shows the various forms of threads. The tables of standard pitches, etc., may be found by looking in the list of tables in this book (page 385).

A great advance has been made in the standardization of dimensional specifications for screw threads. These specifications cover four classes of fits and are applicable to bolts, machine screws, nuts, tapped holes, and other threaded parts. They have been approved and published by the American Standards Association (1935). Incidentally certain terms relating to screws and threads have been changed and others have been added. Except in rare cases the general machinist is not interested in the exact allowances and tolerances,[1] and the lengthy tables[2] are not included in this book. But every machinist should know the *terminology* of threads. Definitions and symbols follow.

Terminology of Threads

188. Definitions.

1. *Screw Thread.*—A ridge of uniform cross section in the form of a helix on the surface of a cylinder or cone.

2. *External Thread.*—A thread on the outside of a member.

3. *Internal Thread.*—A thread on the inside of a member.

4. *Major Diameter.*—The largest diameter of a screw thread. The term major diameter applies to both internal and external threads, and replaces the term "outside diameter" as applied to the thread of a screw and also the term "full diameter" as applied to the thread of a nut.

5. *Minor Diameter.*—The smallest diameter of a screw thread. The term minor diameter applies to both internal and external threads, and replaces the terms "core diameter" and "root diameter" as applied to the thread of a screw and also the term "inside diameter" as applied to the thread of a nut.

[1] For definitions of allowance and tolerance see pages 223 and 224.

[2] For tables see *American Machinist's Handbook.*

6. *Pitch Diameter.*—The diameter of an imaginary cylinder the surface of which would pass through the threads at such points as to make equal the widths of the threads and the widths of the grooves cut by the surface of the cylinder. It is equal to the major diameter less an amount equal to the single depth of thread. The allowance and tolerance in the sizing of threads are given as on the pitch diameter. Also the pitch diameter is used in determining the outside diameter of the *blanks* for rolled threads. As that part of the groove of the thread below the pitch line is rolled (depressed) into the blank the metal is squeezed up to form the part of the thread above the pitch line. Hence the diameter of the blank for rolled threads is equal to the pitch diameter of the screw to be rolled. A great proportion of commercial threads are rolled.

7. *Axis of a Screw.*—The longitudinal central line through the screw.

8. *Pitch.*—The distance from a point on a screw thread to the corresponding point on the next thread measured parallel to the axis.

$$\text{Pitch (in inches)} = \frac{1}{\text{number of threads per inch}}$$

9. *Lead.*—The distance a screw thread advances axially in one turn. On a single-thread screw, the lead and the pitch are identical; on a

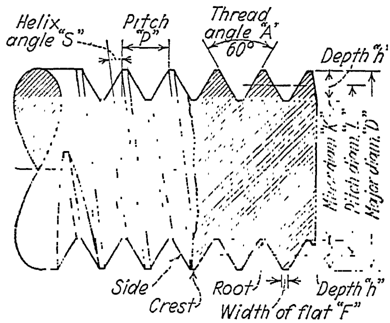


FIG. 170.- –American Standard screw thread.

double-thread screw, the lead is twice the pitch; on a triple-thread screw the lead is three times the pitch, etc.

10. *Angle of Thread.*—The angle included between the sides of the thread measured in an axial plane.

11. *Half Angle.*—The angle included between a side of the thread and the normal to the axis, measured in an axial plane.

12. *Helix Angle.*—The angle made by the helix of the thread at the pitch diameter with a plane perpendicular to the axis.

13. *Crest.*—The top surface joining the two sides of a thread.

14. *Root.*—The bottom surface joining the sides of two adjacent threads.

15. *Side.*—The surface of the thread which connects the crest with the root.

16. *Depth of Thread.*—The distance between the crest and the root of a thread measured normal to the axis.

17. *Length of Engagement.*—The length of contact between two mating parts, measured axially.

18. *Depth of Engagement.*—The depth of thread contact of two mating parts, measured radially.

19. *Thickness of Thread.*—The distance between the adjacent sides of the thread measured along or parallel to the pitch line.

Terms Relating to Fits

20. *Fit.*—The relation between two mating parts with reference to ease of assembly. The quality of fit is dependent upon both the relative size and the quality of finish of the mating surfaces. When any classes of fits are established in machine-shop practice it means that certain "allowances" and "tolerances" are specified.

21. *Allowance.*—An intentional difference in the dimensions of mating parts. It is the minimum clearance or the maximum interference which is intended between mating parts.

Examples.—The following examples are of allowances for three classes, 1, 2, and 4 respectively, of screw and nut ¾ in. diameter, 10 threads per inch, American National Coarse-thread Series.

1. ¾″-10NC-1 (class 1 is the least exacting fit).

Minimum pitch diameter of nut	0.6914
Maximum pitch diameter of screw	0.6850
Allowance (positive)	0.0064

This means that the closest fitting screw and nut, class 1, will have 0.0064″ clearance (space) (play) between sides of threads.

2. ¾″-10NC-2 (class 2 is recommended for general use).

Minimum pitch diameter of nut	0.6850
Maximum pitch diameter of screw	0.6850
Allowance (none)	0.0000

This means that the closest fitting screw and nut, class 2, will have no clearance.

3. ¾″–10NC–4 (class 4 is the most exacting fit).

Minimum pitch diameter of nut	0.6850
Maximum pitch diameter of screw	0.6854
Allowance (negative)	−0.0004

This means that the closest fitting screw and nut, class 4, will be a *tight* fit, with the screw-pitch diameter 0.0004″ larger ("maximum interference") than the pitch diameter of the nut.

22. *Tolerance.*—The amount of variation permitted in the size of a part.

Example.—Screw ¾ in. diameter, 10 threads per inch, American National Coarse-thread Series, class 2 (¾″–10NC–2).

1.	Maximum pitch diameter	0.6850
	Minimum pitch diameter	0.6786
	Tolerance	0.0064
2.	Maximum major diameter	0.7500
	Minimum major diameter	0.7372
	Tolerance	0.0128

NOTE.—In American Standard screw threads, all series and all classes, tolerances are applied *plus* to the threaded hole or nut, *minus* to the screw. Thus the variations from basic will always favor the free fit rather than the tight fit.

23. *Basic.*—The theoretical or nominal standard size from which all variations are made.

24. *Limits.*—The extreme permissible dimensions of a part. In the example given in 22 above, the sizes 0.6850 and 0.6786 in. are the limits.

25*a*. *Clearance in nut at minor diameter* is such that the basic depth of thread shall be reduced by one-sixth (or more in some cases).

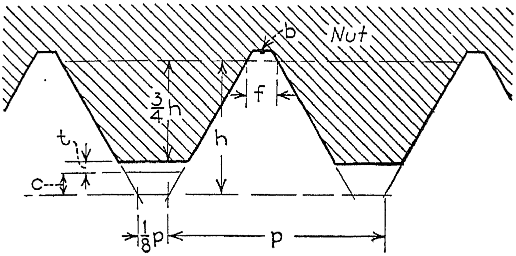
25*b*. *Tolerance in nut at minor diameter* is such that the basic depth of thread may be further reduced by one-twelfth (or more for some sizes of threads).

NOTE.—What 25*a* and 25*b* mean practically is this: For any class of American National form of internal thread (nuts, etc.) the tap-size drill selected may be large enough to leave only three-quarters of a full thread (one-sixth reduced for clearance, plus one-twelfth tolerance, equals one-fourth of thread and leaves three-fourths of full thread) (see Fig. 171 and Table 7, page 392).

26. *Tolerance on major diameter of nut* is such as to result in a flat one-third of the basic flat.

27. *Crest Clearance.*—Defined on a screw form as the space between the crest of the thread and the root of the mating part.

NOTE.—What 26 and 27 mean practically is this: The crest of the tap thread forms (cuts) the major diameter of the nut, and to be sure to have "crest clearance" between the crest of the screw and its mating surface in the nut, the crest of the *tap* is extended. That is, the tap is made oversize on the major diameter, and instead of having the crest equal one-eighth of the pitch it may be only one twenty-fourth of the



n = Number of threads per Inch
h = Basic depth = $.6495 \times p = \frac{.6495}{n}$
c = Clearance = 1/6 of basic depth $\left.\right\}$ = 1/4 of basic depth
t = Tolerance = 1/12 of basic depth
3/4 h = 3/4 of basic depth (or 3/4 full thread in the nut)
b = Extended major diam. on tap to give say 40% truncation instead of 100% truncation as at f

FIG. 171.—Shows portion of a standard nut.

pitch. Taps are usually made with about 40 per cent "truncation" (see *b*, Fig. 171).

For other crest clearance on the screw, see "tolerance major diameter," the second example and the note under definition 22. For crest clearance on nut thread, see definitions 25*a* and 25*b*.

28. *Right-hand and Left-hand Threads.*—A right-hand thread advances clockwise, a left-hand thread counter-clockwise. Looking at the side of a screw, the slant of the right-hand thread is down towards the right, left-hand thread down towards the left.

189. Wire Measurement Symbols.—For method of measuring American Standard threads by three-wire method see page 248.

Measurement over wires	*M*
Diameter of wire	*G*
Corresponding radius (½*G*)	*g*

190. Dimensional Symbols.—For use in formulas, on drawings, etc., the following dimensional symbols should be used:

Major diameter	D
Corresponding radius	d
Minor diameter	K
Corresponding radius	k
Pitch diameter	E
Corresponding radius	e
Angle of thread	A
One-half angle of thread	a
Number of turns per inch	N
Number of threads per inch	n
Lead	$L = \frac{1}{N}$
Pitch	$p = \frac{1}{n}$
Helix angle	s
Tangent of helix angle	$S = \frac{L}{3.14159 \times E}$
Width of basic flat at top, crest, or root	F
Depth of basic truncation	f
Depth of sharp V thread	H
Depth of basic American National thread	h

191. Identification Symbols.—These are for use in correspondence, on drawings, shop and storeroom cards, specifications for parts, taps, dies, tools, and gauges.

The basis of the system is the initial letters of the series, preceded by the diameter in inches (or the screw number if under ¼ in. in diameter) and the number of threads per inch. The initial letters are followed by the classification of fits, all in Arabic characters. If the thread is left hand the symbol "LH" shall follow the class of fit. No symbol is used to distinguish right-hand threads.

Examples	Mark
American Standard Coarse-thread Series: To specify a threaded part 1 in. in diameter, 8 threads per inch, right hand, and class 2 fit	1″–8NC–2
American Standard Fine-thread Series: To specify a threaded part 1 in. in diameter, 14 threads per inch, left-hand thread, and class 3 fit	1″–14NF–3LH

American Standard 8-*Pitch*-, 12 *Pitch*-, *and* 16-*Pitch*-*thread Series:* To specify a threaded part 2 in. in diameter in each of these special series, class 2 fit......... 2″–8N–2 2″–12N–2 2″–16N–2

NOTE.—The number of threads must be indicated in all cases, irrespective of whether it is the standard number of threads for that particular size of threaded part or a special number.

192. The American National Form of Thread.—It is very necessary for the machinist to be able to calculate all the dimensions of the American (National) form of thread, and if he understands the how and why of the calculations relating to this form of thread, he can calculate the dimensions of any thread from the formulas given in handbooks and in most catalogues of drills, taps, etc.

The American Std. thread has an angle of 60 deg. It is in the form of an equilateral triangle (a triangle having equal sides and angles) with the apex flatted. The top of the

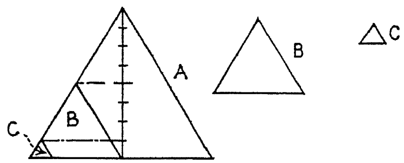


FIG. 172.—Similar triangles have proportional parts.

threads and the bottom of the grooves are not sharp as in the V thread or rounded as in the Whitworth, but are flat. The width of this flat is one-eighth of the pitch of the thread. It will be easier to understand the dimensions of the thread if two or three sketches like Fig. 172 are made.

Draw any two equilateral triangles, the base of one twice as large as the base of the other (for example A, 2-in. base, and B, 1-in. base, Fig. 172). Draw their altitudes (the altitude of a triangle is the perpendicular distance from the apex to the base).

Note that the altitude of B is just one-half the altitude of A. In any two equilateral triangles, the altitudes are proportional to the bases respectively (and of course the bases are proportional to the altitudes). Further to prove

this graphically, draw triangle *C* with altitude equal to one-eighth of the altitude of triangle *A* and note that the base is equal to one-eighth of the base of *A*.

Note, also, that the length of each altitude is about seven-eighths of the base. It is not exactly seven-eighths (0.875) of the base but is exactly 0.866 of the base (in every equilateral triangle the altitude is proportional to the base as 0.866 is to 1.000).

Draw a light line (Fig. 173), 2 in. long, divide it equally into *AB* and *BC*, and using *AB* and *BC* each as a base draw lightly two equiangular triangles. Connect the vertices by dotted line *p*. Draw the altitude and divide it into eight

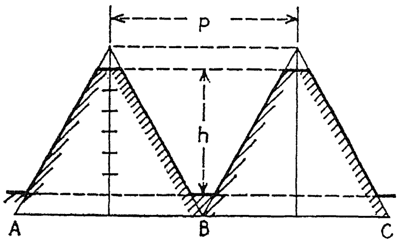


FIG. 173.—Development of American National form of screw thread.

equal parts. Draw light lines through top and bottom dividing lines. Trace heavy line and shade as shown and the figure will represent the profile of two threads with the groove between.

The pitch of the thread is indicated by *p*. It is "the distance from a part of one thread to the corresponding part of the next."

The depth of the thread is indicated by *h*. It is "the perpendicular distance from the crest of the thread to the root of the thread."

The pitch of the thread is equal to the base of the original triangle in the figure, but since one-eighth of the altitude was cut off top and bottom to give the flat top (crest) and bottom (root) of the thread, each equal to one-eighth of the pitch, then the depth *h* of the thread is three-fourths of the altitude of the original triangle.

The altitude of the original triangle is equal to 0.866 of the base, therefore the depth of the thread is equal to three-fourths of 0.866 of the base or 0.6495 of the base. And since the base of the triangle and the pitch of the thread are equal, the depth of the thread is 0.6495 of the pitch of the thread. (From now on the terms triangle, base, and altitude will not be needed; the terms thread, pitch, and depth are understood.)

The *depth* equals $0.6495 \times p$ or, as usually stated, $0.6495p$.

The *double depth* equals $2 \times 0.6495p$ or $1.299p$.

The *minor diameter* equals the major diameter minus $1.299p$.

To multiply by the pitch is equivalent to dividing by the number of threads per inch, therefore, to obtain the minor diameter, subtract from the major diameter the quotient obtained by dividing the constant 1.299 by the number of threads per inch, or expressed as a formula:

Formula for finding exact minor diameter of American Standard screw thread—

$$K = D - (1.299 \div n)$$

Example.—Find the minor diameter of a screw ¾ in. in diameter, 10 threads per inch.

Solution: $750 - (1.299 \div 10) = 0.750 - 0.1299 = 0.620 = K$.

The above formula is very important; be sure to *understand* it.

Questions on Threads

1. In what ways may threads be formed on a screw? In a nut?

2. Name and describe three different forms of threads.

3. What is the difference between the pitch and the lead of a thread?

4. What is the difference between the square thread and the Acme thread?

5. Define "minor diameter" and "double depth."

6. What is meant by "modified V thread"?

7. What is meant by the term "standard" as applied to thread pitches? As applied to thread forms?

8. What are the differences between the American National and the Whitworth Std. forms of thread?

9. What is meant by Acme Standard? A. S. M. E. Std.? S. A. E. Std.? Which are obsolete?

10. State the double depth of an American Std. thread in terms of pitch.

11. What is the formula for determining the minor diameter of an American Std. thread?

12. What is the exact root diameter of a 7/8″–9NC–2 thread?

193. Taps.—Most internal threads are cut with taps, usually in a tapping machine or with a tapping attachment in the drill press. Many threads, however, must be tapped by hand. Figure 174 shows a set of machinist's hand taps squared on the

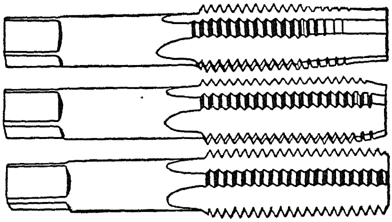


FIG. 174.—Set of hand taps, taper, plug, and bottoming.

shank end to receive the wrench (Fig. 175). Taps are manufactured in all of the standard sizes and pitches for the standard forms of threads. They are generally turned from the solid bar; accurately threaded; carefully and scientifically fluted, that is, grooved to form cutting edges; marked as to

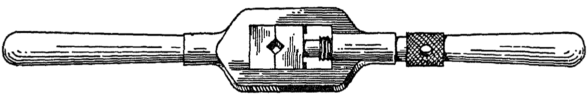


FIG. 175.—Adjustable tap wrench.

size and number of threads per inch; hardened and tempered; possibly ground—taps with ground threads may be purchased at about the same price as cut threads.

194. Tap Sets.—Hand taps, except the sizes under 1/4 in., are made in sets of three taps (see Fig. 174), called taper, plug, and bottoming. The first tap or "taper tap" is tapered or "chamfered" back from the end at least six threads, the plug is chamfered about three or four threads, while the

bottoming tap is merely backed off on the end teeth. Taps furnished in sets are of the same diameter unless otherwise specified, so that to tap a through hole it is only necessary to use the taper tap. Where the hole does not go through the piece ("blind hole") it is customary to start with the taper, follow with the plug, and, occasionally, if the hole is fairly shallow, finish with the bottoming.

Serial hand taps may be used to advantage in tough material. A set consists of three taps. The No. 1 tap, a taper tap, roughs out the thread; the No. 2 tap, a plug tap, being a little larger in pitch diameter, cuts the thread a little fuller; and the No. 3 tap, bottoming, larger than No. 2, finishes the thread to full size.

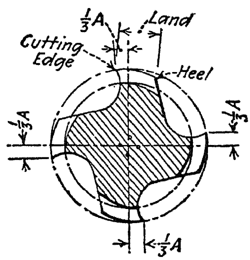


FIG. 176.—Relief of tap.

195. Relief of Taps.—The cutting edges of the chamfered portion of the tap are given clearance, that is, they are "backed off" or "relieved" the whole width of the land, otherwise the tap will not "bite." Further, in order to reduce the friction between the teeth of the tap and the work being tapped, the teeth are relieved about two-thirds the width of the land (Fig. 176). The remaining third of the land back of the cutting edge remains the full cutting size so that the tap may be ground on the face of the teeth several times without changing the size.

196. Tap-size Drills.—The diameter of the hole to be drilled for the threads in a nut or any inside-threaded piece is theoretically the minor diameter of the corresponding screw size. This size of hole will give a *full thread* but it is *not practical* or *customary* to tap a full thread, therefore the *tap-drill sizes* are usually larger than the minor diameter.

Three-quarters of the double depth of thread is enough to leave for tapping. An ordinary nut drilled out so that it has only half of a full depth of thread will break the bolt before it will strip. A two-thirds to three-fourths depth of thread will give a margin of safety of about two to one and requires only

about one-third the power for tapping that is required to tap a full thread. As explained in paragraph 192, the full double depth of an American Std. thread is obtained by dividing 1.299 by the number of threads per inch. Consequently, dividing three-fourths of 1.299 by the number of threads per inch will give three-fourths of the full depth of thread (three-fourths of $1.299 = 0.974$, or 1.000 approximately).

Therefore in ordinary machine work, if a list[1] of tap-drill sizes is not at hand, the size of a hole to be drilled for tapping may be found by subtracting from the outside diameter of the tap, the quotient obtained by dividing 1.000 by the number of threads per inch. Select the size of drill required or the nearest sixty-fourth under for taps over ¼ in. diameter, and the nearest number size for taps ¼ in. and under.

Example.—½-in. tap − 13 threads. *Solution:* $0.500 - 1.000 \div 13 = 0.500 - 0.077 = 0.423$ in. The nearest drill under 0.423 in. is 27/64 in. (0.421 in.).

197. Length of Tapped Hole.—Except for the purpose of holding the screw fixed, as in a tool post for example, it is unnecessary to have the tapped portion of a hole longer than one and one-half times the diameter of the screw. In fact a well-fitting screw entering a tapped hole a distance equal to its diameter will break about as soon as the threads will strip. This fact is often overlooked and deep-tapped holes are called for, which makes for waste of time and breakage of taps.

198. The Operation of Tapping.—A certain pressure is needed to start the taper tap and care must be taken to make the tap "bite" or "catch the thread" and not ream the top of the hole taper. After the tap is well started it feeds itself and requires only to be turned. It is a good plan occasionally to turn the tap backward half a turn to break the chip, and in tapping soft material such as copper, babbitt, etc., it is necessary to remove the tap several times and clean away the chips.

Care must be taken to start the tap square and keep it square. A tapping bushing is a valuable aid. It consists merely of a tapped bushing faced square. If the diameter of

[1] For list of tap-drill sizes see Table 7, page 392.

the bushing is three or four times the diameter of the tap it will effectively prevent the tap from tipping.

Use lard oil as a lubricant when tapping steel, wrought iron, or other metals, except cast iron. Tap cast iron dry (sometimes, however, a little oil or soap put on the teeth that are rubbing in that part of the thread already cut will ease the tap). Many mechanics prefer to use a mixture of turpentine and lard oil when tapping copper and a mixture of kerosene and lard oil when tapping aluminum.

A sharp V-thread tap is an excellent roughing tap where a smooth-tapped hole is required, since it leaves a small amount

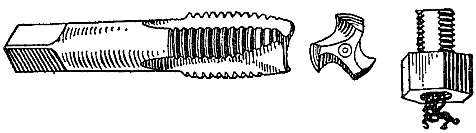


FIG. 177.—"Gun" tap. (*Greenfield Tap and Die Corporation.*)

of stock on the sides of the thread for the standard tap to remove as a finishing cut.

Wrapping a suitable piece of waste over the end of the tap and turning it through, will serve to make the tapped hole slightly larger.

Taps are usually obtained from the toolroom in sets, together with the body-size and tap-size drills, and often with the counterbore for the head of the screw.

The G. T. D. gun tap (Fig. 177) is a very efficient tap for the reasons that it is strong, shears the chip, and does not clog since the chip "shoots" out ahead of the tap.

Cautions.—Take note when resizing or "cleaning" a previously tapped hole that the piece has not been hardened, otherwise the tap may be ruined. For instance do not attempt to retap a case-hardened nut.

Certain manufacturers still use special pitches of threads. For example ½ in.–12 is frequently used for setscrews instead of ½ in.–13. If a standard-pitch screw does not fit easily, determine the pitch of the tapped thread before forcing in the screw.

Questions on Taps and Tapping

1. Name the three taps in a tap set. What is the purpose of each one?

2. How do you make sure that a tap is started square? Why is it necessary?

3. How hard is a tap? Is it brittle? Why does it break easily?

4. What is meant by the "feel" of a tap when cutting?

5. What is an adjustable tap wrench? Solid wrench? What are the advantages of each? What is a disadvantage of an adjustable wrench?

6. What is meant by "clearance" on a tap?

7. Why is it advisable when tapping to frequently turn backwards a quarter or a half-turn?

8. When threads are tapped in tough metal, what should be done to keep them from tearing?

9. What lubricant is best when tapping cast iron? Steel? Brass? Aluminum? Copper?

10. How far should a screw enter a tapped hole in order to give sufficient strength?

11. What is a "blind" hole?

12. How do you use a scale when counting the number of threads per inch? What is a "pitch gauge"?

13. How may the pitch of the thread in a nut be determined with a whittled stick, if there is no tap or bolt to fit it?

14. What is a tap drill? What is meant by a "body-size" drill?

15. Why is a tap drill smaller than the outside diameter of the bolt?

16. How many threads per inch had the 3/4-in. U. S. Std. screw? The 3/4-in. S. A. E. Std. screw?

17. Why are the screws used in automobile work of finer pitch than those used in machine-tool manufacture?

18. In common practice, is the tap-drill size equal to the diameter of the thread? Is it larger or smaller?

19. State three objections to the use of a tap drill that will give a full thread.

20. Calculate the size of the tap drill for 5/8″–11NC–2 screw. Calculate the size of the tap drill for 5/8″–18NF–1 screw.

21. Calculate the size of the tap drill for 7/8″–9NC–1 screw. Calculate the size of the tap drill for 7/8″–14NF–2 screw.

22. If it is desired to tap a slightly larger hole in a nut so it will be a free fit on a thread, how can this be done with a standard-size tap?

23. Before attempting to retap an old nut, what precaution should be taken? Why?

199. The Threading Die.—A *die* is a tool for cutting external threads. In general the threading die is so arranged as to

permit the cutting edges of four cutters or chasers to do an equal share toward cutting their shape (the form of the thread desired) into a cylindrical rod, when turned or "screwed" on the end of the rod for a distance of the required length of the thread.

Some dies are made solid, some in two halves within a body or "head," and still others with the four separate chasers properly and securely held in the head. The last named (see Fig. 178) is perhaps the best type since the chasers can be easily removed and sharpened. It also permits of considerable adjustment which is a decided advantage when a screw slightly oversize or undersize is required, or when a roughing and finishing cut are desirable. The complete die when locked together seems to have all the advantages of a solid die.

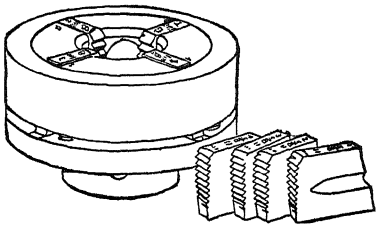


FIG. 178.—Adjustable die (*P. & W.*).

Most threads used in manufacturing are cut with dies in screw machines and bolt machines. Very often, however,

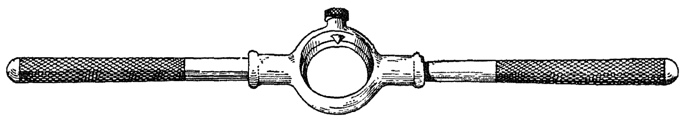


FIG. 179.—Die holder or diestock.

it is practicable to size a fairly long thread or even to cut the whole thread "by hand" in which case the die is held in the *diestock* or *screwplate* (see Fig. 179).

When threading a piece by hand the end of the rod should be chamfered about 45 deg. for at least the depth of the thread and care must be exercised in starting the die true. As in starting a tap, pressure must be exerted when starting the die, but after it is well started it will feed itself. Use lard oil when cutting a thread on steel and wrought metals, and turn backward part of a turn occasionally to break the chip.

Questions on the Use of Threading Dies

1. How can you chamfer the stock with a file? By what other means may it be chamfered?
2. Give two reasons for chamfering.
3. What care must be taken in starting a die? Why is it necessary?
4. Is it better to cut the entire thread in one cut or in two cuts? Why? How do you adjust the die for this?
5. What lubricant is best to use when cutting a thread on steel or wrought iron?
6. Why is it advisable when cutting threads with a die to frequently turn backward a quarter or half turn?
7. When cutting the thread, what should be done to keep it from tearing?
8. How do you protect the thread already cut on one end of a stud when threading the other end?
9. When it is desired to cut the thread close to the head on a bolt or cap screw, how can it be done?
10. What is meant by a die? Die holder? Diestock?
11. What is meant by a solid die? Adjustable die?
12. What advantage has an adjustable die?
13. What special advantages has a die with removable cutters?
14. What precaution should be taken before attempting to recut threads on an old bolt or screw? Why?

GEARING A LATHE FOR CUTTING THREADS

To cut a thread in a lathe involves the use of gears. It is therefore essential that the beginner understand the first principles of spur gearing in order intelligently to set up his machine.

200. Definitions and Explanations of Terms Used.—*A spur gear* is a toothed wheel or cylinder with the teeth parallel to the axis. The smaller of the two gears in mesh is often called the *pinion*.

A train of gears is a series of two or more gears with teeth in mesh. The motion of the first gear causes each gear in the train to move.

A bank of gears is a number of gears arranged together and revolving on or keyed to the same shaft or sleeve. When there is a bank of gears of different sizes arranged successively, the collection is often called a *cone of gears*.

A unit of several gears arranged as sliding gears to give a series of speeds is called a *cluster of gears.*

In a simple gear train, the gear to which motion is first imparted is the *driving gear* and the gear of this train to which motion is *finally* transmitted is the *driven gear,* or *follower.*

In a gear train the speeds of the gears are *inversely proportional* to the number of their teeth. For example: A driving gear has 20 teeth and a follower gear has 40 teeth; one revolution of the driving gear will engage 20 teeth of the follower gear and cause it to make one-half of a revolution, that is, the follower gear, which is *twice as large* as the driving gear will revolve *half as fast.* In the same way the follower gear *half as large* as the driving gear will revolve *twice as fast.* The speeds are not directly proportional to the number of the teeth of the gears, but are indirectly or *inversely* proportional.

A gear in mesh between the driving and follower gears is called the *intermediate.* The purpose of an intermediate gear is to connect two gears that are too far apart to mesh with each other.

In a gear train each gear revolves in the opposite direction to that of the gear with which it meshes, therefore, adding an intermediate gear serves to change the direction of the follower gear. An intermediate of any number of teeth or any number of intermediates may be used, and not change the relative velocities of the driving and follower gears.

Example.—Driving gear 28 teeth, driven gear 28 teeth, one revolution of the driving gear will cause one revolution of the driven gear. Introduce one intermediate of any number of teeth, say 60 teeth, one revolution of the driving gear will engage 28 teeth of the intermediate and it in turn will engage 28 teeth of the follower gear causing it to make one revolution or the same as was obtained without the intermediate. The direction of rotation of the follower gear, however, is changed.

In simple gearing then, the size of the intermediate may be disregarded. The velocity of the driving gear is to the velocity of the follower gear inversely as the numbers of their teeth.

201. Gearing the Lathe for Cutting Threads.—As has been previously stated, the more modern lathes are provided with *quick-change gears* for feeds and thread leads. It is not enough, however, for a machinist to be able merely to move a handle or two to set the gears to cut a given thread, he must know the *reason why.* Perhaps the best way to learn about the lathe gears for thread cutting is to get an understanding of the older method of change gears, a description of which is given here. On pages 51 to 54 a brief discussion of quick-change gears will be found.

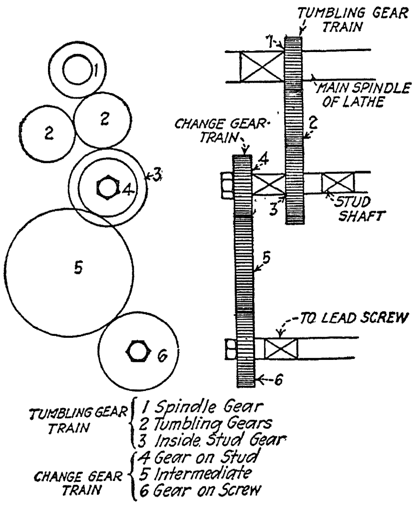


FIG. 180.

Thread cutting in an engine lathe is accomplished by causing the lathe carriage to move, *positively*, a certain distance for each revolution of the main spindle.

The positive movement of the carriage is obtained by, first, connecting the main spindle to the lead screw by gears, thus transmitting any movement of the spindle, positively, to the lead screw; and, second, closing tightly the split nut upon the lead screw thereby ensuring a positive movement of the carriage for each revolution of the lead screw.

When cutting threads in a lathe the motion of the spindle is transmitted to the *stud shaft* by the *tumbler-gear train* and from the *stud shaft* to the *lead screw* by the *change-gear train* (see Fig. 180).

The tumbler-gear train consists of a gear keyed to the spindle, two *tumbler gears* (or *reverse gears*), and the fixed stud gear which is keyed to the inside end of the *stud shaft* (Fig. 180).

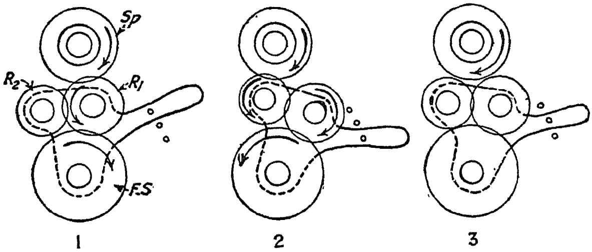


FIG. 181.—Illustrates operation of reverse gears or tumbler gears.

The two tumbler gears are intermediate gears between the spindle gear and the fixed stud gear and are so mounted on a bracket as to make it possible for the operator to have one intermediate or two intermediates in mesh between the driving and driven gears or to throw them both out of mesh with the driving gear. That is, with the three positions of the handle as shown (Fig. 181), the lathe hand may have (1) forward movement, (2) reverse, (3) neutral or no motion of the stud shaft.

The function of the tumbler gears is to reverse the direction of rotation of the feed rod, if for any reason it is desired to feed toward the tailstock, or of the lead screw when cutting left-hand threads. With a forward motion of the spindle and of the lead screw, the carriage advances toward the headstock and a right-hand thread is cut. To cut a left-hand thread, the work turns forward just the same but the direction of the lead screw is reversed thus moving the carriage toward the tailstock.

The *change-gear train* (so called because the driving and follower gears may be changed at the will of the operator) consists of the *gear on stud*, the *intermediate*, and the *gear on screw*

(see Fig. 180). A series of different gears called change gears are furnished with the lathe and by changing the sizes of the gears on stud and screw various velocity ratios between the two may be obtained.

202. Operation of the Gears.—When the spindle gear and the inside stud gear of the tumbler-gear train (Fig. 180) are of the same size, and the gears on stud and screw are equal, one revolution of the stud shaft will cause one revolution of the lead screw. If the lead screw is $\frac{1}{6}$-in. pitch the carriage will advance $\frac{1}{6}$ in. The number of threads per inch which will be cut on the work will be the same as the number of threads per inch on the lead screw.

However, *a great many lathes are made with the inside stud gear larger than the spindle gear.* Suppose the spindle gear has 30 teeth and the inside stud gear has 40 teeth; one revolution of the spindle will cause three-fourths of a revolution of the stud shaft and with equal gears on stud and screw cause three-fourths of a revolution of the lead screw.

If the lead screw is $\frac{1}{6}$-in. pitch, the carriage will advance three-fourths of $\frac{1}{6}$ in. or $\frac{1}{8}$ in. and 8 threads per inch will be cut on the work.

If the inside stud gear is twice as large as the spindle gear and the lead screw has 6 threads per inch, 12 threads per inch will be cut on the work with equal gears on stud and screw.

Lead Number.—The number of threads per inch that are cut with equal gears on stud and screw is the basis of all calculations for change gears for thread cutting and is called the *lead number.* The lead number for any lathe may be found as follows: Find the ratio of the number of turns of the *spindle* to the number of turns of the *stud shaft* (whole numbers) and multiply the number of threads per inch on the lead screw by this ratio. For example: 4 turns of *spindle* to 3 turns of *stud shaft,* 6 threads per inch of lead screw, $6 \times \frac{4}{3} = 8 =$ *lead number.*

Any lathe with a lead number of 8 will cut 8 threads per inch with equal gears on stud and screw, will cut 16 threads per inch if the gear on screw is twice as large as the gear on

stud. Or it will cut 4 threads per inch if the gear on screw is half as large as the gear on stud.

203. Calculating the Sizes of Gears to Cut a Given Thread. The *rule for change gears* may best be stated in the form of an equation:

$$\frac{\text{Lead number}}{\text{No. of threads per inch}} = \frac{\text{gear on stud (driving gear)}}{\text{gear on screw (follower gear)}}$$

or as a formula:

$$\frac{L}{N} \quad \frac{D}{F}$$

By this equation the correct gears to cut any thread, whole or fractional, may be quickly ascertained.

Example.—Lead number 6; threads per inch 10; available gears 20 to 80, progression[1] 4; six-tenths is the ratio of the driving gear to the follower gear (gear on stud to gear on screw).

Solution.—A 6-tooth gear on stud and a 10-tooth gear on screw would cut the thread, but no such gears are available. To multiply both numerator and denominator of a fraction by the same number does not alter the ratio, therefore, multiply by 4 and the result equals

$$\frac{6 \times 4}{10 \times 4} = \frac{24}{40}$$

Use a 24-tooth gear on stud and 40-tooth gear on screw

Another example:

To cut $11\frac{1}{2}$ threads per inch

$$\frac{6}{11\frac{1}{2}} \times \frac{6}{6} = \frac{36}{69}$$

Use a 36-tooth gear on stud and 69-tooth gear on screw.

Note.—A 69-tooth gear is usually furnished with a lathe.

[1] *Change-gear Progression.*—By "progression" in change gears is meant the regular increase in the number of teeth in each succeeding gear in a set of gears. The sizes of the gears increase by a certain number of teeth from the smallest to the largest gear—in the above case by 4 teeth from 20 teeth to 80 teeth.

An index plate is fastened to the side of the lathe to show which change gears to use when cutting threads. However, a specified gear may be mislaid or broken, or a thread not given on the plate may be required. The *thinking* mechanic is *resourceful.*

204. Compound Gearing.—Suppose the smallest gear available is a 24-tooth gear and it is necessary to have the follower gear revolve one-sixth as fast as the driving gear. For example, to cut 36 threads with a lead number of 6. In a simple train this would require a 144-tooth follower gear $\left(\frac{6}{36} = \frac{24}{144}\right)$. If such a gear is not available, or if the center distance between

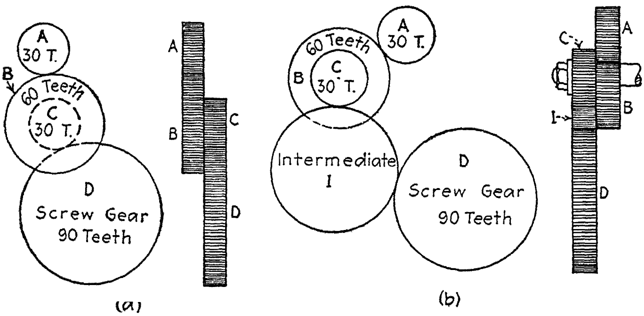


FIG. 182.—Illustrates compound gearing.

the shafts does not permit of its use, an arrangement known as compound gearing may be used to obtain the required result.

In either arrangement (*a* or *b*, Fig. 182) is illustrated a compound of two simple gear trains, *A* the driving gear and *B* the follower gear of the first train, and *C* the driving gear and *D* the follower gear of the second train.

While the compounding gears *B* and *C* are arranged between the original driving gear *A* and the final follower gear *D*, it is evident that their sizes are not disregarded as is an intermediate in a simple train of gears. Suppose gear *A* to revolve 6 r.p.m., *B* would revolve three times, *C* keyed to the same shaft as *B*, would revolve three times and would cause *D* to revolve once because *D* has three times as many teeth as *C*.

The gears B and C are together known as the compound. In many lathes a "compound" is furnished with the machine. It consists of two gears, one having twice as many teeth as the other, fastened together and arranged on a special bracket between the stud gear and the intermediate. This is illustrated in *b*, Fig. 182, the gears B and C forming the compound. In other lathes provision is made to substitute compounding gears for the intermediate as shown in *a*, Fig. 182.

In compound gearing the velocity of the original driving gear is to the velocity of the final follower gear, inversely as the product of the driving gears is to the product of the follower gears. The formula used for a compound train is the same as for a simple train if D equals the *product* of the driving gears and F equals the *product* of the follower gears. Therefore to ascertain the proper gears to use for compound gearing the same rule is used to find the four gears as for finding the two gears in a simple train, namely:

$$\frac{\text{Lead No.}}{\text{No. of threads required}} \quad \frac{\text{driving gears}}{\text{follower gears}}$$

For example: Required to cut 28 threads per inch, lead number 6, progression 4. Arrange the ratio of the lead number to the number of threads to be cut (6:28) as a fraction $\frac{6}{28}$ and factor. $\frac{6}{28} = \frac{2 \times 3}{4 \times 7}$; now multiplying the numerator 2 and the denominator 4 by the same number does not change the value of the fraction and multiplying 3 and 7 by the same number does not change the value of this fraction. Multiplying 2 and 4 by 16 $= \frac{32}{64}$ and multiplying 3 and 7 by 8 equals $\frac{24}{56}$, that is, $\left(\frac{6}{28} = \frac{2 \times 3}{4 \times 7} = \frac{32 \times 24}{64 \times 56}\right)$. Gears 32 and 24 are the driving gears and 64 and 56 are the follower gears. If a compound gear in a 1:2 ratio is furnished with the machine it may be used instead of the 32 and 64 gears. If either the 24 or the 56 gear is not available multiply 3 and 7 by any number which

will give two gears that are available, for example, multiplying by 12 gives 36 driver and 84 follower and these gears may be at hand.

NOTE.—One of the best examples of compound gearing in the machine shop is to be found in the "back gears."

THREAD CUTTING

205. Preliminary Hints on Thread Cutting.

1. Grind the thread tool accurately to gauge (center gauge, Fig. 183).

2. Do not grind the point flat for pitches under 1/6 in., round it slightly with an oilstone.

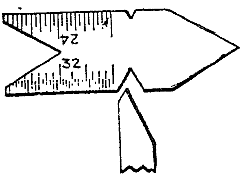


FIG. 183.—Center gauge. A thread tool ground as shown will cut close to a shoulder if desired.

3. Set the compound rest around 30 deg. to the right. This gets the compound-rest handle away from the cross-feed handle and also makes it easy to adjust the tool to catch the thread if necessary.

4. The cutting point of a thread tool should be exactly on center, and the cutting edges set exactly to gauge (see Fig. 184). Do not jam the thread tool into the V of the gauge, but leave a little space between. With the gauge against the work

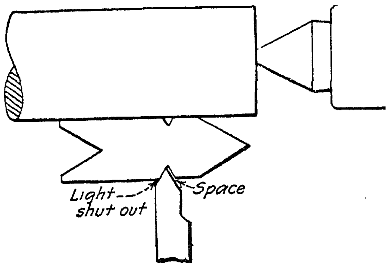


FIG. 184.—Setting the thread tool.

move one side of the little V against one cutting edge of the tool and holding a piece of paper underneath see if the light is shut out. If one side is all right try the other side, and thus check the grinding of the tool.

5. The friction feed is never used when cutting a thread. Be sure the feed-control knob is not tightened. Having the split nut and the feed both in will break the apron when the lathe is started.

6. *Be sure* the gears are right for the pitch required. Many jobs are spoiled by carelessness in this respect. Measure the pitch after the first light cut. This may be done by counting the number for 1 in. or for ½ in. (Fig. 185) but a screw-pitch gauge (Fig. 186) is quicker and there is less chance of error.

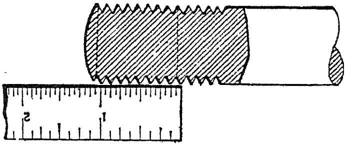


Fig. 185.—Eight threads per inch.

7. Be sure that the split nut is closed *tight* on the lead screw. If the threads on the nut do not go into the grooves of the screw, move the carriage a trifle by hand.

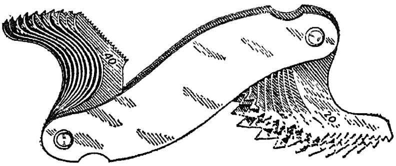


Fig. 186.—Thread-pitch gauge.

8. See if there is plenty of clearance for the dog when the thread tool is at the end of the cut.

9. If the thread being cut is of steel or wrought iron, apply good lard oil or cutting compound with a brush. Do not cut dry, and do not slush the lathe.

10. If there is more than one slot in the faceplate, mark the one used with chalk. If necessary to remove the work from the lathe be sure to put the dog in the marked slot otherwise the thread will be "split" and ruined.

11. Even with a sharp thread tool, a burr will form on top of the thread. This burr should be "brushed off" with a file before the finish cut is taken.

12. If the outside diameter is the correct size the depth of the thread may be gauged fairly close by the amount of the flat left. When approaching the finish, work carefully. It is

often advisable, especially when cutting several screws, to first rough the threads. Finish with a keen tool correctly ground to 60 deg. and take light cuts.

13. To correct a ragged appearance of the first thread, the end of the work should be chamfered. This is done, usually after the thread is finished, by opening the split nut and hand feeding the thread tool carefully against the corner and chamfering it to the depth of the thread.

206. Operation of Cutting the Thread.—The operation of cutting the thread is accomplished by feeding the thread tool in for a light cut, starting the lathe to take the cut for the required distance, backing the tool out of the groove and reversing the lathe to bring the thread tool back to the starting point; feeding in for another light cut (not over 0.003 or 0.005 in.) and repeating these operations until the thread is finished. If the thread tool is allowed to remain in the part of the groove already cut when the direction of the work is reversed, the point of the thread tool will be broken, owing to the backlash in the lead screw and split nut and also in the gearing. It is essential therefore that the thread tool be pulled out of the groove before reversing the lathe. It is a knack to turn back the cross-feed-screw handle and reverse the lathe at the same time, and it might be well for the beginner to practice this for a little while before attempting to cut the thread. Grasp the control handle (or the shipper) with the *right* hand and the cross-feed-screw handle with the *left* hand and practice working the two at the same instant.

207. The Thread Stop.—It is usually advisable for the beginner to use the thread stop (Fig. 187). The thread stop *A* is arranged on the carriage in front of the cross slide and the screw *S* slides freely in a hole in the stop and screws into a hole in the cross slide. Turn the screw *S* until it enters the hole in the cross slide ¼ in. or more. Set the thread tool by the gauge, and on center, and run the cross slide in until the point of the tool nearly touches the work to be threaded. Then by clamping the stop, the cross slide (and tool) cannot be fed in except as the screw *S* is loosened.

In order to feed the thread tool in a certain definite amount for each cut, the screw S is loosened sufficiently to allow the tool to move this amount. The feed, of course, should be less as it approaches the full width of the cut, that is, as it approaches the finish of the thread. Many machinists prefer to use only the graduations of the cross-feed screw for gauging the depth of cut. It will probably save time and trouble,

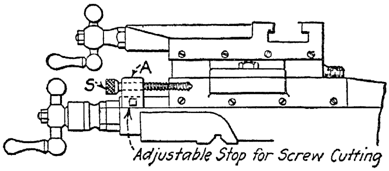


FIG. 187.

especially for the beginner, to use both the thread stop and the graduations.

208. Four Ways of "Catching the Thread" after Resetting the Tool.—If it is necessary for any reason to remove the tool before the thread is finished, *the tool must be reset accurately to gauge,* and then, using one of the following methods, make sure the tool exactly enters the partly cut groove. Be careful to have both the work and the carriage as if going forward rather than reverse, that is, be sure all backlash is taken up.

1. If the lathe is provided with a compound rest, adjust the tool to the desired position in the groove manipulating the cross-feed handle and the compound-rest handle. (See hint 3, page 244.)

2. If the lathe is not provided with a compound rest loosen the dog and turn the work until the tool enters the groove centrally.

3. If this is impractical put the reverse-gear handle in neutral position and revolve the work *forward* by hand, until the tool is exactly opposite the groove, then connect the reverse gears as before.

4. Another way is to disengage the intermediate gear from the screw gear and revolve the spindle forward by hand until the tool will enter the groove of the thread centrally, then reengage the intermediate.

209. Using the Apron-control Handle.—Some lathes are provided with an apron-control lever to start, stop, or reverse the apron without stopping the spindle. This lever, at the right of the apron, serves to move a single-tooth clutch between the forward and reverse gears within the headstock. (This clutch is shown in *a*, Fig. 30, page 53.)

The fact that the clutch is single tooth keeps the revolutions of the stud gear exactly timed with the revolutions of the spindle, and therefore the thread tool will exactly enter the groove to take succeeding cuts. If there were two teeth instead of one on the clutch member, the thread tool would just as likely as not exactly split the thread instead of enter the groove.

210. The Three-wire Method of Measuring Threads.—The value of a micrometer for measuring any piece of work is that the measurement can be *read* at once in thousandths of an inch or less. The piece is not merely "scant" or "full" or "good enough" as the old-time machinist used to say, but it is a certain *number of thousandths* too small, or too large, or it is right. The value of the "three-wire method" of measuring threads is exactly the same. After calculating the size that the measurement over the wires ought to be, it is easy to measure over the wires *with a micrometer* and tell at once how much too large the thread is or when it is the right size.

The three wires are of the same diameter but a definite size for a given pitch is not required; any diameter, smaller than the pitch of the thread, that will not fall below the top of the thread will answer (see G in Fig. 188). Ordinary drill rod, approximately 0.6 of the pitch in diameter, carefully measured to have $3G$ accurate, is satisfactory in most work.[1]

[1] It is true that in allowances and tolerances the pitch diameter has assumed an important position in the measurement of threads, nevertheless it would be difficult to prove that a wire bearing on the thread

Select three wires an inch or two longer than the diameter of the thread. Place two of the wires in adjacent grooves on one side of the screw and one wire directly opposite. Snap a rubber band over the three ends of the wires on each side of the screw to hold them while the measurement with the micrometer is being made. (Some machinists prefer to hold the three wires in a small block of wood as shown in the figure.)

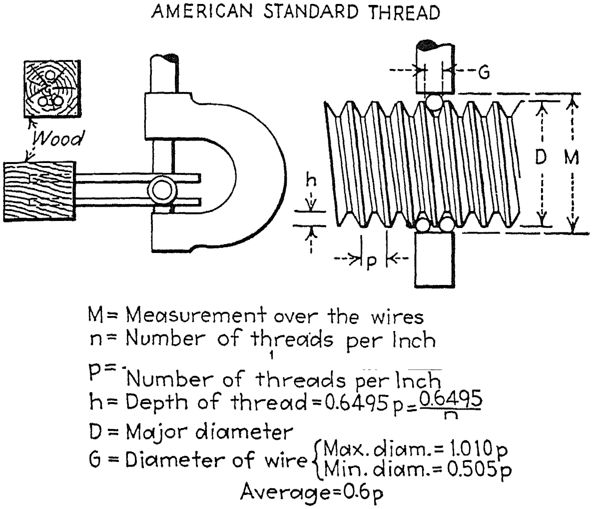


FIG. 188.—Measuring threads by the three-wire method.

To tell whether the thread is cut deep enough, or for the purpose of accurately gauging the thread at any time for depth of cut, the following rule may be used:

211. Measuring the American National Form of Thread.- To the diameter of the screw add three times the diameter of

---

exactly at the pitch diameter is the "best wire." If the thread is made 60 deg. it is as right to measure it at one part as another, if more or less than 60 deg. it is inaccurate anyway.

Further, to calculate sizes of wires to hundred thousandths or even ten thousandths of an inch is unnecessary because the three-wire method is not meant for such a high degree of accuracy. The machinist's common sense must be used many times when using decimals to the third place or further.

the wire and from the sum subtract the quotient obtained by dividing 1.5155 by the number of threads per inch. The measurement over the wires should equal this result.

Formula: $$M = D + 3G - \frac{1.5155^*}{n}$$

Example.—It is required to measure a ⅞″–9 thread.

Solution.—1. Select the wires, say 0.066 in. in diameter which is between the minimum and maximum sizes that may be used.

2. Make the necessary calculation as follows:

$$D = 7/8 = 0.875$$
$$3G = 3 \times 0.066 = 0.198$$
$$\frac{1.5155}{n} = \frac{1.5155}{9} = 0.168$$

Then $0.875 + 0.198 - 0.168 = 0.905$.

3. Placing the wires over the thread, make the measurement over the wires carefully. If this measurement is 0.905 in. the thread is right. If it is greater than 0.905, then it must be made just that much smaller. If it is less than 0.905, the thread has been cut undersize.

Adjust the micrometer very carefully over the wires. Hold it square across the diameter of the thread, and adjust until nearly right. Then hold the anvil of the micrometer tight against the two wires on the bottom, and slightly wiggling the spindle turn the spindle down, easily, until there is no shake.

212. Measuring the V Thread.—In the V thread, instead of using the constant $1.5155/n$ the constant $1.732/n$ is used.

213. Measuring the Whitworth Thread.—In the Whitworth form of thread the constant $1.6008/n$ is used, and for "three times the diameter of the wire" in the rule, substitute "3.1657 times the diameter of the wire.

214. The Use of the Compound Rest for Cutting Threads.—The great disadvantage of the thread tool cutting on both sides of the 60-deg. angle is the fact that it cannot have rake and cut correctly. If a thread tool which is supposed to cut an equal

* For derivation of constants 1.5155 and $3G$ see page 383.

amount on each side of the angle is given rake from one side, the other side will automatically be given a *negative* rake and will not cut. The objection to front rake is shown in Fig. 189. It will be observed that any angle of front rake decreases the angle between the cutting faces, and this angle grows smaller as the rake is increased.

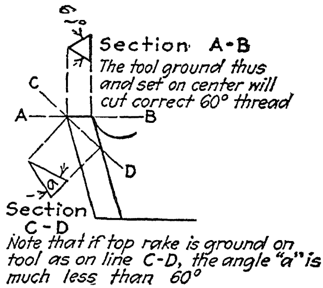


FIG. 189.

If, however, the lathe is provided with a compound rest, a tool having side rake as shown in *a*, Fig. 190, may be used.

Set the compound rest so that the tool may be fed in at an angle of 30 deg. to form one side of the thread. Grind the tool

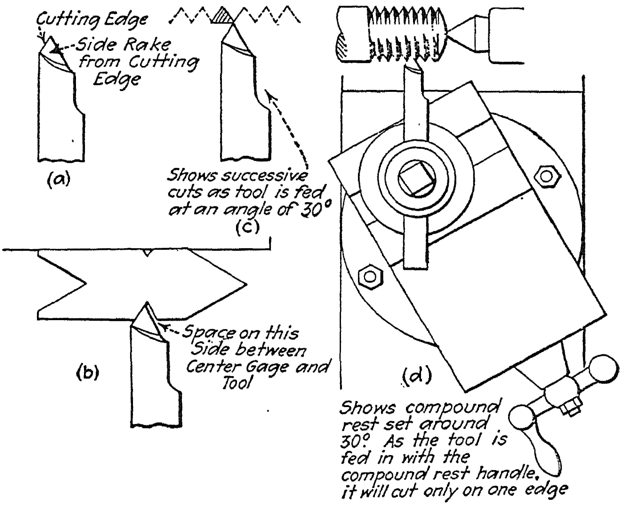


FIG. 190.

to 60 deg. and set it with the center gauge as usual, *b*, Fig. 190. The setup is shown at *d*.

The thread stop is used as a stop only and not to gauge the depth of the cut because the tool is fed into the work by the

compound-rest handle. One side of the tool does all of the cutting (*c*, Fig. 190) and may be given the desired rake. The adjacent side, if the tool is properly ground, will just clear the other side of the thread. That is, the tool is pulled out of the groove at the end of the cut and fed back to the stop just before the start of the next cut, but the successive cuts to make the groove deeper and deeper are fed in by moving the compound-rest handle.

This method of cutting threads is recommended as being two or three times as fast as with a tool having no rake.

215. Cutting a Thread without the Reverse Belt.—Sometimes when cutting a long thread the time wasted in the return

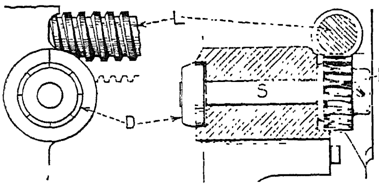


FIG 191.—Thread chasing dial. *D*, dial; *L*, lead screw; *P*, pinion engaging lead screw. In some lathes the position of the shaft *S* is horizontal as above, in others it is vertical. The operation is the same.

of the tool from the end of the thread is a serious consideration, so serious that some shop foremen forbid the use of the reverse belt[1] when thread cutting.

To cut a thread without the reverse belt, the tool is withdrawn at the end of the thread, as is the usual practice, but, instead of reversing the lathe, the split nut is opened and the carriage run back by hand a certain *definite distance*, at which point the split nut may be closed and the operator be assured that the tool will track in the original groove. This definite distance depends of course on the length of the thread but it also depends on the pitch of the thread being cut, and on the pitch of the lead screw.

Many lathes are now equipped with a chasing dial (see Fig. 191, also Fig. 33, page 58). A brass plate giving directions for

[1] *Reverse Belt.*—Sometimes called the "back belt" and often the "cross belt." It reverses the direction of rotation of the countershaft and consequently of the machine spindle.

using the chasing dial on the particular lathe is screwed to the lathe. The dial (being connected with a small gear which meshes into the threads of the lead screw) is caused to revolve once for every 4 in. travel of the carriage. The dial is graduated into eight divisions, consequently each division will indicate travel of the carriage of ½ in. and show at a glance the time when the lead screw and carriage bear exactly the same relative positions as before at which time the split nut is closed and the thread tool will track.

If the lathe has no chasing dial, the place to close the split nut may be determined as follows:

1. If the number of threads required is the same as the number of threads on the lead screw, close the split nut at any point.

2. If the number of threads on the lead screw is a factor of the number of threads required, close the split nut at any point.

3. For all other even threads, close the split nut at any ½-in. distance from the stopping point, if the number of threads on the screw is even; if lead screw is odd close the split nut on any inch distance from the stopping point.

4. For all odd threads close the split nut on any inch distance from the stopping point.

5. For half threads (for example 11½ threads per inch), close the split nut any 2-in. distance from the stopping point.

It is advisable to mark the definite distance from the stopping point of the carriage, perhaps with a lead pencil on the ways of the lathe. Then when the carriage is run back by hand to this mark the split nut will properly engage the lead screw and the thread tool will track. This operation is often spoken of as "catching the thread."

216. To Cut a Left-hand Thread.—To cut a left-hand thread it is necessary to reverse the direction of rotation of the lead screw. This causes the carriage to move toward the tailstock with a forward motion of the spindle. When cutting a left-hand thread start the cut on the end of the thread nearest the dog (usually in a groove already turned) and cut toward the tailstock.

217. To Cut a Thread on a Taper.—When cutting a thread on a tapering piece, for example, on a pipe, the thread tool should be set square with the *center line* of the piece to be threaded. The taper attachment is best and, if available, should be used; if one is not available, and the piece is provided with centers, the tailstock may be offset to give the desired amount of taper. If the work must be held in a chuck and the lathe is not provided with a taper attachment a fairly good job can be done by slowly feeding the tool towards the operator as the work turns.

218. The Square Thread.—For the purpose of transmitting motion an Acme thread or a square thread is nearly always used, as for example on a lead screw or on a feed screw. Also these threads are much used for obtaining and maintaining pressure as on vise screws and jackscrews.

If one has learned to cut an American Standard thread intelligently, he should have no great difficulty in learning quickly to cut a square thread. That is, if it is understood that the centers must be true and in line; that the tool must be the right shape with sharp cutting edges, set accurately to gauge and on center; that there must be room for the carriage to travel without interference from the beginning to the end of the thread; that the gears must be right for the given pitch of thread; that the depth of the cut is important; and that smoothness of the finished thread is very necessary, then nine-tenths of the art of cutting any kind of thread has been learned.

The chief difference in cutting a 60-deg. thread and a square thread is in the tool, although it should be added that to cut a square thread probably calls for patience, carefulness, and strict attention to a somewhat greater degree.

219. The Square-thread Tool.—Although the square-thread tool (*a*, Fig. 192) looks something like a short cutting-off tool it differs in one very important respect: the blade is not square with the bottom as in a cutting-off tool but is canted to conform to the "slant"[1] of the thread. This is illustrated in *b*,

[1] The reason that the thread slants is because it is a helix, and the amount the thread slants depends on the helix angle, "the angle made

Fig. 192. Before attempting to make a tool for cutting a square thread one should know how to determine the correct amount of slant of the tool for the given thread.

The amount the tool slants depends upon two things: (1) The slant angle changes for each different *lead* of thread on a given diameter. The greater the lead the greater the angle. (2) It changes for each different diameter of thread of any given lead. The larger the diameter the less the slant.

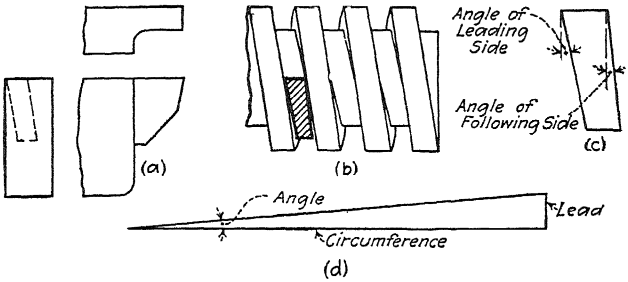


FIG. 192.—The square-thread tool.

In addition to the slant which the tool blade must have it must be made thinner toward the bottom, otherwise it could not enter the thread at all, let alone have clearance, because a piece with parallel sides cannot fit in a curved slot, and the groove of a square thread is a curved slot. The slant of the leading side of the tool therefore must be greater than the slant of the following side of the tool. But notice that both slant in the same general direction (*c*, Fig. 192).

The amount of the slant of either side for any thread may be represented by a right triangle (*d*, Fig. 192). One of the right-angle sides equals the *lead* of the thread and the other the *circumference*, (1) of the minor diameter of the thread for the leading side, and (2) of the major diameter of the thread

by the helix of the thread at the pitch diameter with a plane perpendicular to the axis."

for the following side. The number of degrees of slant is measured between the hypotenuse and the side representing the circumference (*d*, Fig. 192). (The triangle may be drawn to larger scale if desired.)

Example.—Find the slant of the following side and of the leading side of the blade of a tool for cutting 1¼ in.–4 square threads.

Solution:

Lead equals 0.250 in.

Major diameter = 1.250 in.

Minor diameter = 1.000 in.

1.250 in. × 3.14 = 3.92 in. = circumference (major diameter of thread).

1.000 in. × 3.14 = 3.14 in. = circumference (minor diameter of thread).

Draw a right triangle, one right-angle side equal to 0.250 in. (lead) and the other equal to 3.92 in. (circumference of major diameter); draw the hypotenuse and measure the angle between the circumference line and the hypotenuse. It will equal 3⅔ deg. as nearly as can be measured with a bevel protractor. This is the angle of the following side. Draw another right triangle with L (lead) equal to 0.250 in. and C (circumference) equal to 3.14 in. (circumference at the minor diameter of thread); draw the hypotenuse and measure the angle. It will equal 4½ deg. as nearly as can be measured with the protractor. This is the angle of the leading side.

Another way of finding the slant angles of a square thread tool is by means of a simple calculation if a table of tangents is at hand. On the next page the above example is solved by this method. The portion of the table of tangents given is sufficient for any square thread.

Clearance.—Theoretically a cutting-off tool need have no side clearance but for practical purposes it must be given clearance, say 1 deg. on each side, or it will rub. In the same way a square-thread tool must have clearance; *in addition to its theoretical shape* it must be made still thinner at the bottom, say about 1 deg. on each side (see *b*, Fig. 192). Suppose the

Mathematical Method of Finding Slant Angles of Square-thread Tools

In any right-angle triangle, for example *d*, Fig. 192, the *side opposite* divided by *side adjacent* equals the *tangent of the angle.* Therefore dividing the lead (side opposite) by the circumference (side adjacent) will give the tangent of the angle to be found. In the example given (page 256), find the slant angle of the following side. Lead is 0.250, circumference of major diameter is 3.92 in. Then $0.250 \div 3.92 = 0.0637$. Looking in the table below, the nearest number to 0.0637 is 0.0641 which is the tangent of the angle 3 deg. 40 min., the same angle as was found by drawing the figure and measuring with a protractor. (For square threads the angle within a third of a degree is near enough; the table shows every 10 minutes or one-sixth degree.)

Find the slant angle of the leading side. Lead is 0.250, circumference of minor diameter is 3.14 in. Then $0.250 \div 3.14 = 0.0796$. Looking in the table below, the nearest number to 0.0796 is 0.0787 which is the tangent of 4 deg. 30 min.

Table of Tangents up to 14 Deg. 50 Min.

Angles	Tan	Angles	Tan	Angles	Tan	Angles	Tan	Angles	Tan
0° 00′	.0000	3° 00′	.0524	6° 00′	.1051	9° 00′	.1584	12° 00′	.2126
10	.0029	10	.0553	10	.1080	10	.1614	10	.2156
20	.0058	20	.0582	20	.1110	20	.1644	20	.2186
30	.0087	30	.0612	30	.1139	30	.1673	30	.2217
40	.0116	40	.0641	40	.1169	40	.1703	40	.2247
50	.0145	50	.0670	50	.1198	50	.1733	50	.2278
1° 00′	.0175	4° 00′	.0699	7° 00′	.1228	10° 00′	.1763	13° 00′	.2309
10	.0204	10	.0729	10	.1257	10	.1793	10	.2339
20	.0233	20	.0758	20	.1287	20	.1823	20	.2370
30	.0262	30	.0787	30	.1317	30	.1853	30	.2401
40	.0291	40	.0816	40	.1346	40	.1883	40	.2432
50	.0320	50	.0846	50	.1376	50	.1914	50	.2462
2° 00′	.0349	5° 00′	.0875	8° 00′	.1405	11° 00′	.1944	14° 00′	.2493
10	.0378	10	.0904	10	.1435	10	.1974	10	.2524
20	.0407	20	.0934	20	.1465	20	.2004	20	.2555
30	.0437	30	.0963	30	.1495	30	.2035	30	.2586
40	.0466	40	.0992	40	.1524	40	.2065	40	.2617
50	.0495	50	.1022	50	.1554	50	.2095	50	.2648

square-thread tool in the example just solved is given 1 deg. clearance on each side, then the angle to grind on the leading side will be the calculated angle *plus* 1 deg. (4½ deg. plus 1 deg. equals 5½ deg.), and on the following side the angle will be 1 deg. *less* than the calculated theoretical angle (3⅔ deg. *minus* 1 deg. equals 2⅔ deg.). This is illustrated in Fig. 193. *It is difficult at first to reason* why the clearance on a square-thread tool is plus on one side and minus on the other but it is very necessary to understand it. It is a typical machine-shop problem.

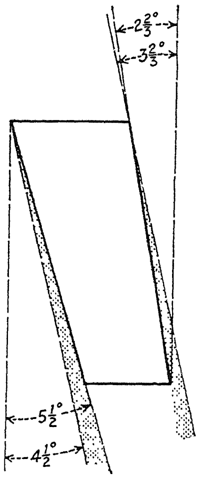


FIG. 193.—Showing the *theoretical* and the *clearance* angles for a 1¼ in.—4 square thread.

Suggestions.—Have the tool a trifle wider than half the pitch, say 0.003 in. on a quarter-inch pitch thread (tool, 0.125 plus 0.003 in.).

Most machinists prefer to grind the tool in a surface grinder or in some other exact way rather than by hand.

If impracticable to grind it then file the annealed tool to shape and harden and temper it.

If many threads are to be cut, it will be advisable to set a blade of the right thickness in a toolholder having a slot milled to fit the blade and at an angle to conform to the slant of the thread. The part of the blade that projects from the holder may be given clearance by grinding a little on each side, making it a little thinner at the bottom. Do not, however, change the thickness of the top of the tool, that is, the width of the cutting edge.

220. Cutting a Square Thread.—The operation of cutting a square thread differs in no particular respect from cutting an American Std. thread. If the thread is half-inch pitch or greater it is usually advisable to cut it somewhat narrower with a "stocking" tool, before finishing. Some mechanics prefer to finish the sides of the thread with a side tool, others prefer the regular square-thread tool ground to full size. In any

event the object is to secure a thread with smooth sides. Proceed as follows:

1. Set the work in the lathe. Be sure the centers are in line. Tighten the dog on the work securely, using a protecting piece of soft brass or copper, and adjust the work fairly tight between centers with plenty of oil (check up on this adjustment occasionally because the strain of the cut tends to enlarge the dead-center hole).

2. Set the compound rest at an angle, say 30 deg., to the right. This is to get it out of the way of the cross-feed handle, and to be able to use it later if necessary to catch the thread.

3. Set the lathe for the lead of thread desired, and oil the lead screw and the ways.

4. Set the tool to the left of the tool rest, on center and square.

5. Put in the split nut and "cut air" to the end of the thread. This is to make sure that there is room for the travel of the carriage to the end of the thread, and also that the thread will end exactly in the center of the hole, if a hole has been drilled for this purpose. If necessary move both the compound-rest feed and the cross feed until the thread ends exactly in the hole.

6. Run the tool in to touch the work, note the graduation, and calculate what the graduation should read when the thread is cut the full depth (depth equals one-half pitch plus three or four thousandths for clearance).

Frequently square- and Acme-threaded screws are designed to permit the end to be turned for a distance of 1/16 in. or more to the minor diameter size. This helps in determining when the thread has been cut full depth. The tool as set for the thread may be used to turn this short shoulder distance.

7. If the length of the thread warrants it, use the chasing dial (page 252).

8. Proceed to cut the thread full depth and fit to a nut or gauge. The feed-in for each cut will depend upon the size of the thread, say 0.005 to 0.010 in. for a 1/4-in. pitch. Use lard oil or cutting compound.

221. The Square-thread Tap.—A tap may be used to finish the inside thread, or several taps, each succeeding tap being larger, should be used to cut the smaller threads, especially when fairly long in proportion to diameter.

When making a square-thread tap the thread is somewhat wider than the groove in order that the screw that goes into the tapped hole will have a trifle clearance on the side. Also the diameter of the tap should be a few thousandths oversize to prevent the outside of the screw thread from rubbing in the tapped hole. Further, the groove should be cut a trifle deeper than the root diameter because this part of the tap does not cut since the hole for the thread is bored at least full root-diameter size. Great care must be taken to back off the sides and top of a square-thread tap clear to the cutting edge, but only a very little. If too much clearance is given a tap, chips will wedge between the tap and the thread when turning the tap backward thus scoring and perhaps spoiling the thread.

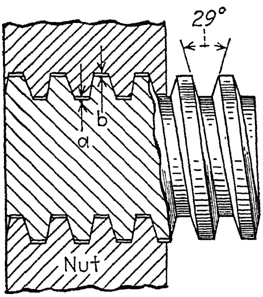


FIG. 194.—Acme screw in nut. *a*, clearance of 0.010 in. is obtained by boring the hole 0.020 in. larger than the major diameter of the screw; *b*, clearance of 0.010 in. obtained by making the major diameter of the tap 0.020 in. larger than the major diameter of the screw.

222. The Acme Thread.[1]—The Acme thread (Fig. 194) is intended to take the place of the square thread because (1) it is easier to cut Acme threads with taps and dies, and (2) the Acme thread is stronger.

It is important to know about the standard clearance in Acme threads. The bearing between any screw and nut is on the *sides* of the threads, and clearance is provided between crest and root, in American National form, square, Acme, and other shapes. The "tap-drill size," that is, the hole in the nut, is made large enough and the major diameter of the tap is made enough oversize to take care of the clearance.[2]

[1] Table of Acme threads, page 396, Acme taps, page 397.

[2] See 25 and 26, pages 224–225.

The square thread, external and internal, has sides parallel and the depth is equal to one-half the pitch, plus a small amount for clearance (no standard amount).

The Acme thread has sides forming an angle of 29 deg., and the normal or working depth is equal to one-half the pitch. The clearance, for both crest and root, is 0.010 in. *for all sizes of Acme threads.* To get this, the major diameter of taps, for all pitches of Acme threads, are made 0.020 in. oversize to give 0.010 in. major-diameter clearance in nuts and other internal threads, and Acme thread tools are properly shaped[1] (to gauge) and the screws are cut 0.010 in. deeper (than normal) to give minor-diameter clearance in the threads.

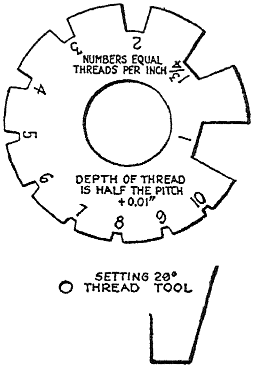


FIG. 195.—Gauges for grinding and setting Acme-thread tools.

Having these clearances provided in the tap and in the screw, the minor diameter of the Acme nut or other internal Acme thread is the normal minor diameter

$$\left(D - \frac{1}{n}\right).$$

223. Cutting an Acme Thread.

1. The Acme-thread tool, external and internal, is ground to gauge for the pitch required (see Fig. 195).

2. The screw thread is cut 0.010 in. deeper than half the pitch on all sizes of threads. That is, minor diameter always equals

$$D - \left(\frac{1}{n} + 0.020 \text{ in.}\right).$$

3. The tap is turned 0.020 in. over major diameter of screw. It is cut with the same width of tool as the screw thread, and the minor diameter is the same for tap and screw.

[1] The end of the tap thread and the end of the thread tool are extended 0.010 in. and are consequently narrower than the normal shape of the thread.

4. The hole for the nut or other internal Acme thread is made a size equal to the major diameter minus an amount equal to the pitch $\left(D - \frac{1}{n}\right)$. This minor diameter will have clearance since the screw thread is cut 0.010 in. deeper than half the pitch.

5. The side of an Acme thread is 14½ deg., therefore move the compound rest 14½ deg. to the right when cutting a screw, to the left when boring an internal thread. Set the thread stop and feed for successive cuts by moving the compound-rest handle.

6. The procedure for cutting a square thread may be followed when cutting an Acme thread (see page 258).

224. Boring an Internal Thread.—Cutting an internal thread of whatever shape is to cutting an external thread as boring is to turning—a little more care for the spring of the tool, a little more difficulty in measuring. The compound rest is moved around to the *left* instead of the right, and the tool (cross slide) is moved towards the operator for the cut, away from the operator before reversing. Make the setup carefully, follow directions when cutting the first thread and the second will be easy.

1. Internal threads are best finished with a tap, but if no tap is available the thread must be bored to size. If the piece to be fitted is heavy or awkward to handle it will save time and trouble if a cheap gauge is made before the thread is started. To make such a gauge, take careful measurement of the thread to be fitted and cut a short screw of the same diameter and pitch.

2. It will be wise for the beginner to make a sketch of the internal thread to be cut. Give the sizes of the major diameter, depth of thread, and minor diameter, in thousandths of an inch. Except in special cases, the hole for *American National form of thread* may be larger than the exact minor diameter by an amount equal to one-half the single depth of thread (one-half of 0.6495 in. $\times$ p). For a *square thread* the hole should be bored a trifle (0.005 in.) over the minor diam-

eter (square-thread minor diameter equals major diameter of screw minus an amount equal to the pitch). For example: Minor diameter for $1\frac{1}{8}$ in.–4 square thread equals 1.125 minus 0.250 equals 0.875. The hole will be bored about 0.005 larger or 0.880. For the *Acme* thread the hole is bored 0.020 in. over the minor diameter. To obtain the size to bore the

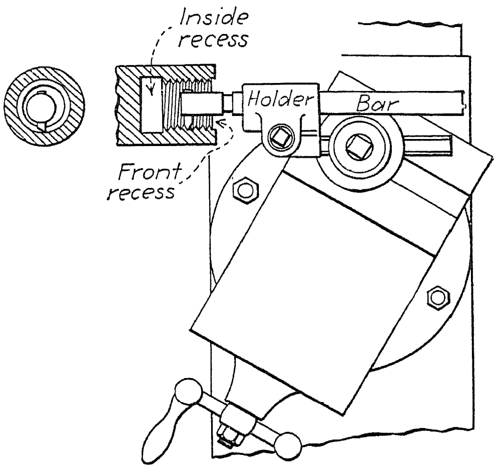


FIG. 196—Showing setup for boring a thread. The compound rest is set around to the left 30 deg. for the American Standard thread, $14\frac{1}{2}$ deg. for an Acme thread. Set the thread stop as explained in the text (9, page 264) and use the compound-rest handle for feeding successive cuts.

hole for an Acme thread, subtract an amount equal to the pitch from the major diameter of the thread (see page 261).

3. Nearly always, it is necessary to bore the hole to a certain size before cutting the thread, first to have the hole run true; second, to give it the correct size. It may be advisable to have two tool bits and possibly two complete boring-tool holders, one for boring, the other for threading. An excellent boring-tool holder (with the bit set at right angles to the bar, as required for cutting threads) is shown top view, in Fig. 148, page 189.

4. If the thread is not to go all the way through, an inside recess should be bored. This recess (Fig. 196) is a groove at

least as deep as and somewhat wider than the thread, for the thread to run into.

5. It will be easier to bore the thread if the tool has rake, as explained in paragraph 214, page 250, therefore set the compound rest around 30 deg. as shown in Fig. 196. It is better to set the compound rest around anyway, (1) to get the handle away from the cross-feed handle, (2) to make it easier to reset the tool if necessary. (For a square thread, set the compound rest around, but *feed* with the cross-feed handle. For an Acme thread, set the compound rest around 14½ deg. and feed with the compound-rest handle.)

6. The thread tool is carefully ground with special attention to clearance and sharpness.

7. To prevent unnecessary spring, the bar of the tool should be reasonably large in diameter and caught as short in its holder as the length of the thread will permit. Set the thread tool on center.

8. In most cases it will be permissible, before proceeding to cut the thread, to bore a slight front recess, equal to the major diameter, to act as a gauge for the depth of the thread. This small shoulder may be bored with the thread tool after it is set for threading.

9. Set the thread stop to act in a double capacity, in one direction to limit the size of the cut, in the opposite direction to prevent the tool from being run too far back, in which event it will rub off the top of the thread.

10. It will be advisable to have some mark, on the bar perhaps, that will indicate the point of reversal.

11. Have the setup right, proceed carefully, use cutting oil for steel. The depth of successive cuts will depend upon conditions. The cut with 60-deg. thread tool is merely a deep line at first—wider and wider as the finish is approached, but less depth of cut on account of the increasing width.

225. Cutting Metric Screw Threads.—A meter, the standard of length in the metric system of measurement, is equal to 39.37 inches.

For purposes of finer measurement the meter is divided into 100 equal parts called centimeters and the centimeter into 10 equal parts called millimeters.

Therefore a millimeter is equal to .03937 inch ($1/1000$ of a meter or $1/1000$ of 39.37 inches). Also if 39.37 inches (one meter) equals 1000 millimeters *one inch equals* 25.4 *millimeters* (1000 millimeters divided by 39.37).

In all threads except the metric it is customary to speak of the number of threads per inch, that is, the thread of $1/13$-in. pitch for example, is thought of and spoken of not as a "$1/13$ pitch" thread but as "13 threads per inch." In the metric thread, however, it is the usual practice to think and speak in terms of pitch and the pitch of the thread is given in millimeters. [See list of tables: French (Metric) Standard Threads and International Standard Threads.]

For example the 26-mm. thread (1.024 in. diameter) has a pitch of 3 mm. (0.118 in.).

In order to determine the number of threads per inch of a metric thread it is necessary to divide 25.4 by the pitch in millimeters. That is, $\frac{25.4}{\text{pitch (in mm.)}}$ is the number of threads per inch.

The formula for gearing a lathe to cut any desired number of threads per inch is:

$$\frac{\text{Lead number}}{\text{No. of threads per inch required}} = \frac{\text{driving gears}}{\text{follower gears}}$$

Substituting Pitch (mm.) for the "number of threads per inch required" in the above formula and

$$\frac{\text{Lead number}}{\dfrac{25.4}{\text{pitch (in mm.)}}} = \frac{\text{driving gears}}{\text{follower gears}}$$

Suppose the lead number is 8 and the pitch is 3 mm., then:

$$\frac{8}{\dfrac{25.4}{3}} = 8 \div \frac{25.4}{3} = 8 \times \frac{3}{25.4} = \frac{24}{25.4}$$

The denominator being fractional, both terms of the fraction are multiplied by 5 (in this case) to get a whole number.

$$\frac{24 \times 5}{25.4 \times 5} = \frac{120}{127} = \frac{\text{driving gear (stud)}}{\text{follower gear (screw)}}$$

Now 127 is a prime number, it cannot be factored, so it is impossible to cut a metric thread on a lathe with an English-measure lead screw without a 127-tooth gear, and, further, this gear is always a *driven* or *follower* gear. It is called a "translating" gear.

In the above example 120 stud gear and 127 screw gear will serve, but few lathes are equipped with a gear of 120 teeth.

It will be necessary to compound.

$$\frac{120}{127} = \frac{60 \times 2}{127 \times 1} = \frac{60 \times 40}{127 \times 20} = \frac{\text{60 and 40 (driving gears)}}{\text{127 and 20 (follower gears)}}$$

Some lathes are provided with compounding gears in the ratio of $^{50}/_{127}$ especially for cutting metric threads. In such lathes the gears on stud and screw may be figured to conform to this compounding ratio. Thus in the above example:

Factoring $^{120}/_{127}$ to obtain 50 in the numerator will give

$$\frac{120}{127} = \frac{50 \times 2.4}{127 \times 1} = \frac{50 \times 96 \text{ (driving gear)}}{127 \times 40 \text{ (follower gear)}}$$

or

$$\frac{50 \times 48 \text{ (driving gear)}}{127 \times 20 \text{ (follower gear)}}$$

or

$$\frac{50 \times 72 \text{ (driving gear)}}{127 \times 30 \text{ (follower gear)}}$$

226. Multiple Threads.—In a single thread the lead is equal to the pitch.

A double thread is one having two thread pitches to one lead; a triple thread has three pitches to one lead; a quadruple thread has four pitches to one lead, etc. These threads are known as multiple threads and are much used in machine

construction and may be of any recognized form of thread (see *a*, Fig. 197, which represents a square thread).

Suppose it is required to operate a part of a machine by screw action; that the movement must be ¼ in. per revolution of the screw, and that the diameter of the screw cannot be over 1 in. A single-cut screw of 1 in. diameter ¼-in. pitch will look like *b*, Fig. 197, which is not a good-looking screw and

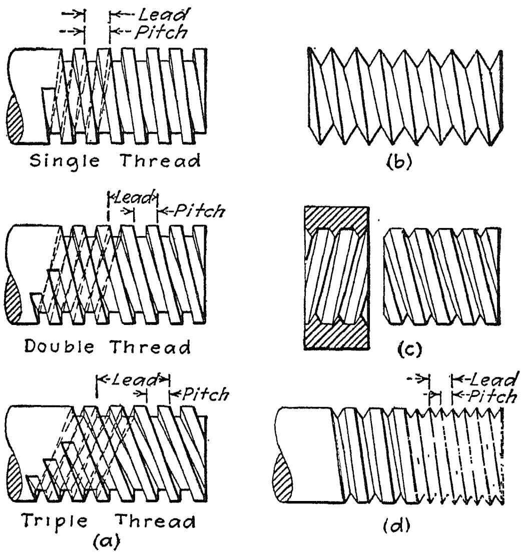


FIG. 197.—Multiple threads.

moreover the cross section at the root of the thread is proportionally weak. If a thread of the same *lead* but half the depth were cut it would look like *c*, still faulty in appearance and the nut would be very weak, in fact only half as strong as it should be.

If a thread of the same depth as in *c*, is cut halfway between the grooves of the first thread as shown in *d*, the *double thread* will be pleasing in appearance and of the required strength. It is, of course, understood that the nut also must have the double thread.

227. Cutting a Multiple Thread.—To cut a double thread, American Std. for example, proceed as if cutting a single thread of the *required lead* until the thread is half the depth, and the groove is half the width of a single thread *of the same lead*. It is then necessary to give the work exactly half a turn without turning the lead screw. This may be accomplished by having a special faceplate with the slot for the tail of the dog exactly opposite the one used for the thread groove already cut.

In the absence of an accurately slotted faceplate, the method used is to disengage the intermediate gear from the screw gear and move the lathe spindle (and the work) one half turn. Before disconnecting these gears bring a tooth of the stud gear exactly between two teeth of the intermediate and mark this tooth with chalk. The distance this tooth moves shows how much the spindle has moved and here is where a serious mistake is liable to occur.

If the gear on spindle and the inside stud gear are of the same number of teeth, the marked stud gear will move one-half revolution when the work is turned half around, but most lathes are now constructed with the spindle gear smaller than the inside stud gear in a ratio of 3:4 or 2:3. This means that the stud gear will not go half around; it will go three-fourths or two-thirds of half around as the case may be.

Suppose the spindle gear has 30 teeth and the inside stud gear has 40 teeth, the ratio is 3:4. Instead of the marked stud gear revolving half around it will revolve three-fourths of one-half revolution or three-eighths revolution. For this reason it will be necessary to select a stud gear with a number of teeth divisible by 8.

Beginning with the tooth *next* to the marked tooth count (in the proper direction) the number of teeth necessary to show the half revolution of the spindle and mark the last one. Turn the spindle to bring that tooth into exactly the proper position with respect to the intermediate gear and engage the screw gear.

The principle of cutting triple threads and quadruple threads is the same as for cutting double threads.

Questions on Cutting a Thread in a Lathe

1. How is thread cutting accomplished in an engine lathe?

2. What is the purpose of the tumbler gears?

3. Why are there a number of change gears furnished with the lathe?

4. With equal gears on stud and screw the lathe will cut a certain number of threads per inch. This is called the *lead number*. The lead number may or may not be the same as the number of threads per inch on the lead screw; give the reason for this.

5. State a rule or formula in the form of a proportion that will serve for calculating change gears.

6. If the lead number of a lathe is 8, what gears are suitable to cut 12 threads per inch? 10 threads? 6 threads? 11½ threads?

7. In any lathe, does it make any particular difference how many teeth the intermediate gear has? Give reason.

8. Why is the intermediate gear adjustable on the bracket? Why is the bracket adjustable?

9. What is meant by compound gearing?

10. When is compound gearing used in thread cutting?

11. With a lead number of 6, gear progression 4, what gears may be used to cut 36 threads per inch?

12. What is the object of having a flat on the top of the thread? On the bottom?

13. What is the angle of a thread tool (American Std.)?

14. What gauge is used when grinding the thread tool?

15. How much clearance has a thread tool? Which side? Why?

16. Why is the point rounded slightly instead of being flattened an exact amount? How is it rounded?

17. If a thread tool is to cut on both sides of the angle, can it be given rake? Give reason.

18. Why is it wrong to pinch the gauge between the tool and the work? How should the gauge be used?

19. What is the purpose of the thread stop? How is it arranged?

20. Why is it necessary to withdraw the thread tool at the end of the thread before reversing?

21. Why do you use a lubricant? How is it applied?

22. How much of a chip is advisable when cutting a thread? Why not more?

23. How do you judge when the thread is nearly cut?

24. How do you remove the burr from the thread?

25. Which is the better method of gauging a thread, with a nut or by the "three-wire method"?

26. When gauging with a nut, can you tell exactly how much more you have to cut? Can you tell with the "three-wire method"?

27. Why do you use three wires instead of two?

28. Why is a $\frac{1}{32}$-in. wire too small for $\frac{1}{10}$-in. pitch thread? Why is $\frac{1}{4}$-in. wire too large? How do you judge the size?

29. State the rule for measuring American Standard threads by the "three-wire method."

30. Why is the "three-wire method" particularly valuable if a special tap is to be made?

31. If for any reason the tool is removed before the thread is finished, what care must be taken when resetting it?

32. State two ways of resetting the tool central with the part of the thread already cut.

33. When resetting the tool why must the lost motion of the lead screw be taken into consideration?

34. What is meant by change-gear progression in a lathe?

35. Is it advisable to use the compound rest when cutting a thread? Give reasons.

36. Explain the setup for using the compound rest for thread cutting.

37. Explain the way in which a left-hand thread is cut in a lathe.

38. What is the advantage of the chasing dial on the apron of the lathe?

39. Is the chasing dial necessary when cutting threads without the reverse belt? Explain.

40. What gear is necessary when it is desired to cut a metric thread with a lead screw of $\frac{1}{6}$-in. pitch? Why?

41. When is a double thread used?

42. How is a double thread cut in a lathe?

CHAPTER XI

FACEPLATE WORK

A large variety of jobs that cannot be machined on centers or held in a chuck may be fastened to the large faceplate[1] and turned or bored.

It is of course essential that the work is accurately fastened or "clamped" in position on the faceplate and for this pur-

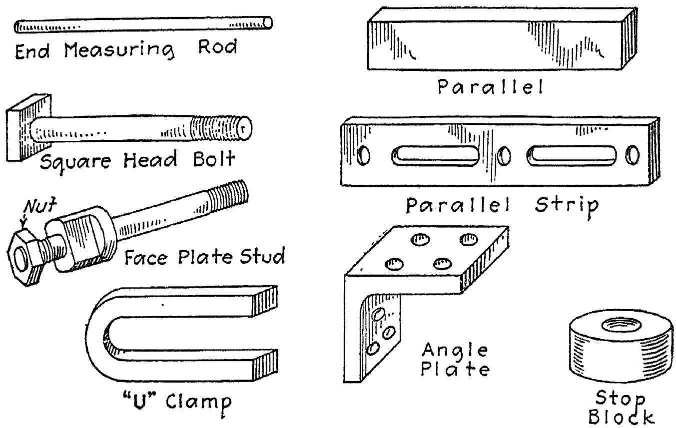


FIG. 198.—Accessories for faceplate work

pose certain accessories are necessary. Sketches and descriptions of some of these are here given (Fig. 198), and examples showing their use follow.

228. Definitions of Accessories Used (Fig. 198). *Square Head Bolt.*—May be used in any of the faceplate slots for clamping pieces to the faceplate.

[1] *Large Faceplate.*—Screws on the threaded nose of the spindle: is as large as will conveniently swing over the ways of the lathe; faced flat and true; has, usually, four or more radial T slots and several shorter slots which go through the plate.

Shouldered Stud.—Threaded each end and so designed that it may be fastened in any desired position on the faceplate.

"U" Clamp.—Used with either a bolt or a stud in clamping the work, is easily adjustable, and is light and strong.

Parallel Strip.—Made with at least two adjacent sides straight and square; is provided with slots and either straight or tapped holes as shown in the figure, so that it may be easily clamped by bolts or screws in any position on the faceplate.

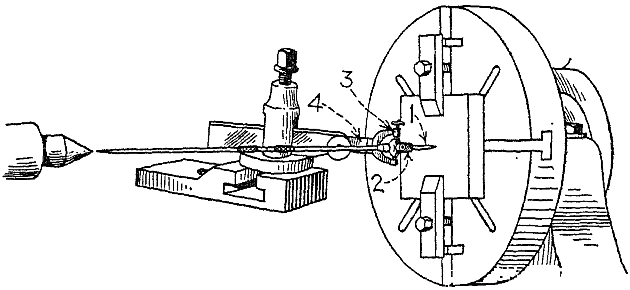


FIG. 199.—Center tester or "wiggler" (*L. S. Starrett Company*). The needle (1) is adjustable lengthwise in ball chuck (2). The ball chuck is pivoted to form a universal joint (3) when indicating a prick punch mark. It may be converted to a single pivot and a 3/16 in. steel ball slipped over the point of the needle for indicating holes or buttons. The flexible steel ribbon (4) keeps the even pressure of the needle or ball against the work.

Angle Plate.—Made in various sizes usually with an angle of 90 deg. between the finished faces, but may have any desired angle; provided with the necessary holes for clamping purposes; is used for a large variety of jobs in faceplate work.

Stop Block.—A small piece of iron or steel, with a hole through it, so that it may be bolted securely to the faceplate forming a positive stop in relocating a piece, or locating several duplicate pieces in the same position on the faceplate.

End Measuring Rod.—Sometimes called pin gauge, made from a piece of drill rod or similar material in any desired length; rounded sufficiently on the ends to offer a point contact and used for relocating work at certain definite distances from the stop block.

Parallel.—A standard shop tool, opposite sides parallel, adjacent sides square, very useful in faceplate work when used in connection with the parallel strip.

Indicator.—There are several different forms of universal indicators by means of which work may be accurately located either from a cylindrical projection on the work, or a hole in the work, or a prick punch mark in the work (see Figs. 199 and 200, also Fig. 94, p. 138).

Weights for Counterbalance.—Very necessary in faceplate work. Any piece of iron or steel that may be picked up in

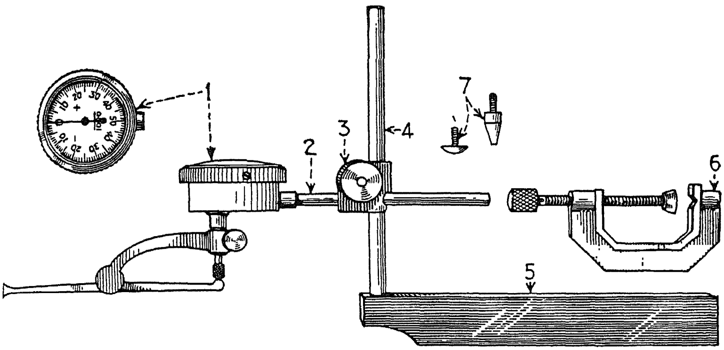


FIG. 200.—Universal dial indicator (*Brown & Sharpe Mfg. Company*) The advantage of a dial indicator is that the error can be read in thousandths of an inch. 1. Dial gauge. 2. Rod, fits hole in dial gauge. 3. Sliding swivel, adjustable on rod (2) or on upright (4) which is fastened in the bar (5). 6. Clamp. Hole in end fits rod (2). With either the bar (5) or the clamp (6) it is possible to use this tool in any machine for almost any indicator testing purpose.

the shop that is of sufficient weight to counterbalance the work when fastened to the faceplate may be used.

229. Typical Faceplate Setups.—Figure 201 shows a flat piece of cast iron clamped to the faceplate located by means of a parallel strip and stop block. This piece has been indicated to drill and bore a hole at one end. With the work in the position shown it would be out of balance were it not for the weight used as a counterbalance.

Figure 202 shows this same piece of iron moved along on the parallel strip and relocated by means of this strip and the

gauge between it and the stop block. With the proper number of lengths of gauges a series of holes at any distance apart may be bored in such a piece, and by using a parallel strip for locating the edge, these holes will be in line.

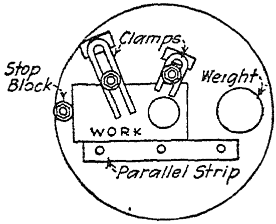


Fig. 201.

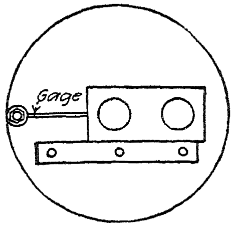


Fig. 202.

Figure 203 shows the use of a parallel by means of which two rows of holes may be drilled in parallel lines; the weights, clamps, etc., are not shown. These operations are used in toolmaking, that is, in making jigs, fixtures, gauges, etc.[1]

Sometimes when it is required to hold on the faceplate and machine a number of duplicate pieces the faceplate may be

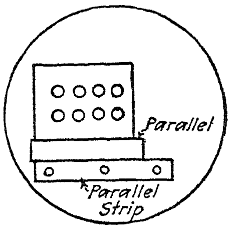


Fig. 203.

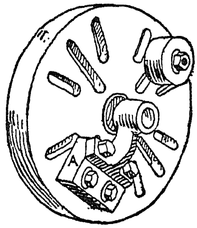


Fig. 204.

set up as a temporary fixture by using parallel strips, or angle irons, or stop blocks, or possibly all three, together of course

[1] Instead of using the parallel as shown "size blocks" may be used. Size blocks are rectangular pieces of hardened steel with two opposite sides ground and lapped to an exact dimension. Size blocks may be used also instead of the pin gauge for spacing the holes. The most accurate of these gauges are the Johansson gauge blocks or "Jo-blocks," which measure accurately in millionths of an inch. They are made by the Ford Motor Company. It will be understood that these gauges are used only where extreme accuracy is necessary.

with the necessary clamps. Figure 204 shows a typical faceplace job of this kind. The hole in the work must be bored at an angle of 65 deg. with the finished flat surface. The special block *A* is planed at an angle of 25 deg. (complement of 65 deg.) to give the required seat for the work. After the block is once set it is not disturbed and duplicate pieces may be quickly and accurately located and machined.

Figure 205 will give an idea of how, in a number of duplicate pieces, the hole may be bored accurately, a certain distance from a finished surface, by locating each piece with the finished surface against a parallel strip, and then clamping to the faceplate as shown.

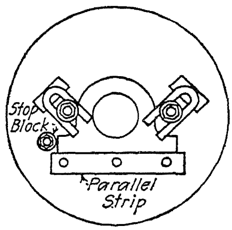


FIG. 205.

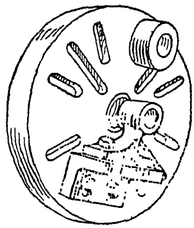


FIG. 206.

Figure 206 shows the value of the angle plate in faceplate work. The angle plate must be square and true. It is usually held against the faceplate by screws coming through from the back of the faceplate and screwing into tapped holes in the angle plate, though often straight holes are drilled through the angle plate and it may then be bolted fast to the faceplate. It is often necessary in order to get the angle plate in the position desired on the faceplate, to drill new holes. In such a case a certain amount of common sense should be exercised in regard to the position of these new holes.

230. Hints on Faceplate Work.—1. Be sure that both the shoulder on the spindle and the face of the hub of the plate are free from burrs or nicks, and that both threads are clean.

2. The faceplate should screw freely on the spindle and tight against the shoulder. It should not be forced too hard

against the shoulder or it will jam. If a chuck or a faceplate is run against the shoulder with a bang, removal is difficult.

3. The faceplate should run perfectly true and if advisable may be tested with an indicator.

4. After the piece is clamped try every screw, and every nut, to make sure that each is sufficiently tight.

5. See that the faceplate is free to turn, that no bolts or clamps project in any way that will come in contact with either the carriage, the ways, or the headstock. Turn once around by hand to make sure.

6. A piece of paper between two flat surfaces will reduce the tendency to slip. This is true in planer work, or shaper work, or boring-mill work, and especially true in faceplate work, where facilities for clamping are not always of the best.

7. Remember in clamping, that it is the work that is to be clamped, not the blocking under the other end of the clamp.

8. Use the dead center to hold the work against the faceplate while clamping. If the work has a large hole in it a piece of flat stock slightly larger than the hole may be used between the work and the dead center.

9. The work and necessary counterweights, etc., may often be more easily clamped to the plate when it is lying in a horizontal position on the bench. The clamps may be tightened sufficiently to hold the work, after which the faceplate is mounted in its place, and then the work carefully adjusted to the desired position and made fast.

10. Use bolts long enough to obtain the full strength of the nut, or the thread of both will be strained and may be spoiled.

11. Avoid using bolts that are much too long; it is dangerous. If other bolts are not available put all excess of length possible back of the faceplate.

231. The Button Method of Locating Holes.—Toolmaker's buttons, *a*, Fig. 207, are steel bushings which may be tightened in exact positions on the work for locating purposes. The hole in the button is larger than the screw to permit of a certain amount of sidewise adjustment of the button after preliminary fastening. The buttons are hardened. They are ground to a

given diameter in even tenths, 0.300 or 0.400 or 0.500 in., and the ends ground square at the same time. The hole is usually $\frac{3}{16}$ in. and the screw 5–40NC–2, that is, $\frac{1}{8}$ in. in diameter.

The method of using the buttons is: (1) Set all the buttons on the work in the exact positions the holes are to be bored; (2) set the work on the faceplate with the No. 1 button trued with the indicator; (3) remove this button, drill and bore the hole, ream if advisable; (4) readjust the work and true up each button in turn and proceed as in (3). The whole procedure in detail follows.

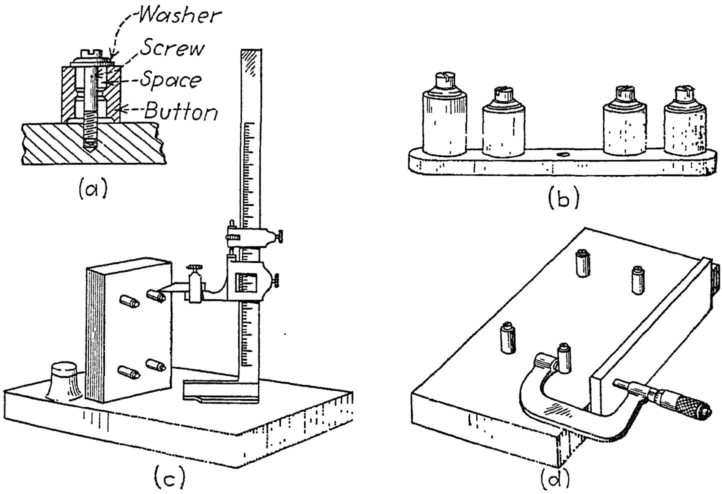


FIG 207.—Toolmaker's buttons. (*a*) Button in place. (*b*) The set of four (*B. & S. Mfg. Co.*) one longer than the rest to permit of indicating when close to another. (*c*) Setting with height gauge (see Fig. 217). (*d*) Setting with parallel and micrometer.

232. Setting the Buttons.

1. The work must be surfaced flat and true.

2. Make a careful layout of intersecting lines to indicate the centers of the holes.

3. Make prick punch marks at intersections, and *check*.

4. Make larger indentations with center punch, and drill for tap (for 5–40 tap, No. 38 drill is used, $\frac{1}{4}$ to $\frac{5}{16}$ in. deep).

5. Tap the holes and file off the burrs.

6. Fasten the No. 1 button (only fairly tight so that later it may be adjusted, that is, rapped this way or that with comparative ease).

7. Adjust the No. 1 button in right relation to the base and edge of work. Use micrometer and parallel, or height gauge.

8. *Tighten* the No. 1 button, and *check*.

9. Proceed in the same way to adjust and tighten the No. 2 button, then No. 3, and so on; each in its proper relation to the base and to the side or to other buttons as the case may be.

233. Setting the Work.

1. Being careful not to disturb the buttons, clamp the work on the faceplate approximately in position.

2. Mount the faceplate on the lathe spindle.

3. Adjust the *work* until the No. 1 button runs true as shown by the indicator.

4. Remove the button, drill, bore, and ream the hole (if the previous work has been done carefully the small tap-drill hole should not run out enough to hurt).

5. Move the work and adjust it until No. 2 hole runs true, and proceed as with No. 1. The same with remaining holes.

Questions on Faceplate Work

1. What precautions are taken when the large faceplate is put on the spindle?

2. What operations may be done on work which is fastened to the faceplate?

3. What tools are used to fasten work to the faceplate?

4. What is a test indicator?

5. What weight is often necessary when work is fastened to the faceplate? Why?

6. Why should a piece of paper between the work and the faceplate serve to help hold the work?

7. How is a clamp arranged so that the work is held most securely?

8. What is an angle plate? A parallel? A shouldered stud?

9. How is a pin gauge made? Why is it rounded on the ends?

10. For what kind of work is the faceplate valuable?

11. Why is the hole in the button larger than the diameter of the screw?

12. Why are buttons ground on the ends?

13. Why is it necessary to make a fairly accurate layout for the tapped holes for locating the buttons?

14. What are "Jo-blocks"?

BENCH WORK

CHAPTER XII

HAMMERS, SCREW DRIVERS, WRENCHES, HACK SAWS

234. **The Use of Hand Tools.**—There are many operations in machine-shop work which involve the use of tools that are controlled by hand. The term "bench work" is used in reference to the operations incident to the processes of laying out, fitting, assembling, etc., when the work is placed on the bench or in a bench vise, and the term "floor work" applies to the larger work which is erected on the floor of the shop. The same tools—hammers, wrenches, cutting tools, measuring tools, etc.—are used for either.

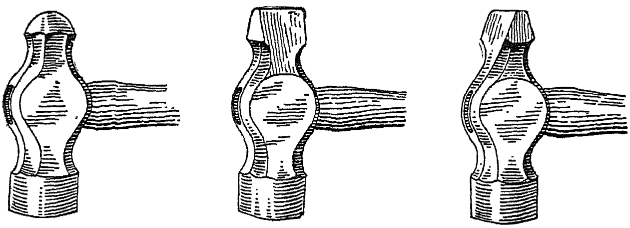
An expert machinist is skillful in the use of the hand tools of the craft. Being skillful is the opposite of being awkward or clumsy; it is the opposite of being ignorant or stupid; it is the opposite of being careless or indifferent. You would not use a tack hammer to drive a spike; why should you use a fine file for roughing stock? You would not use a sledge hammer to drive a tack, why should you use a 12-in. monkey wrench on a ¼-in. bolt? You do not mistake a chisel for a screw driver, why should you use a screw driver with the end shaped like a dull chisel?

Skill means the knowledge of the proper tool to use and how to use it in the right way; it means more than this—it means a positive unwillingness to use a tool that is not right. The real machinist is proud of his kit of tools, proud of the work he can accomplish with them. Manual skill must be acquired through practice. Information regarding the proper use of hand tools can, however, be obtained by reading and by observation.

Bench work involves the use of hammers, screw drivers, wrenches, hack saws, chisels, files, scrapers, taps, threading

dies, small drills (by means of "hand drills" or "breast drills"), hand reamers, taper reamers, and taper-pin reamers. In addition most of the machine-shop measuring tools and gauges are used in bench work. Many of these tools and gauges have been described elsewhere in this book. Also the use of taps, dies, and hand-operated reamers while perhaps belonging particularly to bench or floor work has been previously described and descriptions of these tools will not be repeated here.

235. Hammers.—Machinists' hammers are made of steel, hardened and tempered. They are made in different sizes (weights) from 6 oz. to $2\frac{1}{2}$ lb.; those weighing over $1\frac{1}{4}$ lb.



(a) (b) (c)

FIG. 208.—Hammers. (a) Ball peen. (b) Straight peen. (c) Cross peen.

are not much used. The top of the hammer head is called the peen and the bottom is called the face. The illustration, *a*, Fig. 208, shows a hammer with a "ball" peen which is the common form of machinist's hammer, although the straight peen *b* and the cross peen *c* are much used for swaging and riveting. The eye of a hammer head is somewhat smaller in the middle than at the ends. If the end of the handle is fitted to fill one end of the eye, and wedged with soft steel wedges to fill the other end, it will be tight and secure in the hammer head. *A hammer with a loose head is dangerous.* The handle is set about square with the head and should be of such a shape and length as to give the proper grip and balance when in use.

If one is using a sledge it is much more effective to take it easy when lifting the sledge and put the snap in the blow.

The same is true in all hammering; a solid snappy blow is more effective and less tiresome. Such a blow cannot be delivered if the handle is grasped too near the head or is grasped too tightly.

A hammer blow is a terrific force when applied carelessly. Remember that cast iron is brittle and breaks easily. Do not use a heavy blow on a weak section or on an unsupported projection. Use care when driving a taper pin or key. Remember that the work is softer than the hammer and if the hammer slips and strikes a finished surface the dent made cannot be removed.

Soft Hammers.—Hammers made of lead or soft babbitt are used instead of steel hammers to seat work in a machine vise, or drive a mandrel or arbor, or in any similar operation where the steel hammer might injure the work. A piece of lead more or less spherical in shape about the size of a baseball makes an excellent soft hammer.

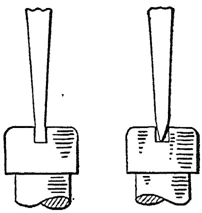


FIG. 209.

236. Screw Drivers.—The screw-driver blade is made of tool steel of the size and length desired. Those under ½-in. cross section are usually made of round stock while the larger sizes are better if made square so that a wrench may be applied if necessary. One end of the blade is forged to provide a suitable tang for the handle, and the other end is drawn out and flattened to fit the slot of the screw. This end of the screw driver is hardened and the temper drawn to a purple verging into a blue (about 550°F.).

It is safe to say that most screw drivers are incorrectly shaped (Fig. 209); the flat sides taper more or less all the way to the end and consequently do not bear against the parallel sides of the screw slot. The reason for making screw drivers in this way is no doubt to produce a tool that will work in a variety of widths of screw slots. If this is true, it is too often carried to extremes with the result that most of the used screw slots are mutilated. According to the principle of the action

of an inclined plane a "wedge-shaped" point on a screw driver will tend to come out of the slot when turned. If considerable twisting force is exerted and not enough end pressure, the screw driver will "jump" out of the slot and throw up an ugly burr. This spoils the appearance of the screw and irritates whoever is unlucky enough to rub his finger across it. A correctly made screw driver is one of the little things that stamp a real mechanic.

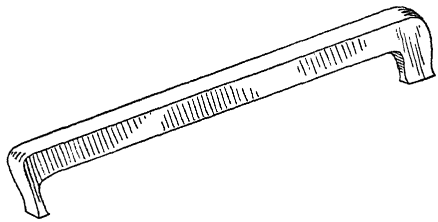


FIG. 210.—Double-end offset screw driver

A double-end offset screw driver (Fig. 210) is used for driving screws that cannot be reached with a straight screw driver.

Questions on the Use of Screw Drivers

1. What happens if considerable force is exerted and the screw driver jumps out of the slot?

2. Is the slot in a screw made with parallel sides or is it V shaped?

3. Does a dull cold chisel make a good screw driver? Give reason.

4. If the screw-driver blade is ground to look like a dull cold chisel will it work well?

5. How is a burr thrown up on the side of the screw slot? Does filing it off fix the slot? Is it fair to leave it on? What should be done?

6. Is there any real reason for applying two forces (holding down and turning) when using a screw driver?

7. Give three reasons why a screw-driver blade should have parallel sides.

8. Of what kind of steel is a screw driver made? Why?

9. How is the tang end shaped to hold fast in the handle? What other methods of holding may be used?

10. Why are the larger sizes of screw drivers often made with square blades (or bodies)?

11. How is a screw driver hardened?

12. What care must be taken when heating? Why?
13. How far back is the blade hardened?
14. What is the proper temper color for a screw driver? Is it harder or softer than a file? Why?
15. What is a double-end offset screw driver? When is it used?

237. Wrenches.—Wrenches are made in many different forms for turning (twisting) bolts, nuts, pipes, taps, etc. Any form of wrench consists of a handle (or handles) with jaws, lugs, or an opening or socket to fit the object to be turned. They are named (1) from their shape; as "S" wrench, angle wrench, etc. (2) From the object on which they are used; as tap wrench, pipe wrench, etc. (3) From their construction; as spanner wrench, ratchet wrench, etc. Several of the wrenches commonly used in machine work are illustrated in Fig. 211.

238. A Few Suggestions Regarding the Use of Wrenches.

1. The wrench should fit, otherwise the corners of the bolt or nut to be turned will be rounded. The wrench is a lever and the mechanical advantage is of course in proportion to the length of the handle. The handle of a solid wrench is usually made to give about all the leverage the part to be turned will stand without injury. In an adjustable wrench this cannot be the case. When a large monkey wrench is used on a small bolt, or when a large tap wrench is used on a small tap or reamer, considerable care and judgment must be used. With the extra leverage the workman is not so sensitive to the resistance of the bolt or tap and may turn too hard and break it.

2. Do not use a wrench with an opening too large. When using a monkey wrench adjust the sliding jaw until it is tight on the nut.

3. When using a monkey wrench, have the jaws point in the direction the force is to be applied. There are two reasons for this; the wrench is less apt to be sprung, and it is less liable to slip off the nut or bolt.

4. A quick jerk when tightening, or a blow with the ball of the hand when loosening a bolt or nut, is more effective than a sustained pull or push, because momentum is a factor.

5. It is distinctly *not* good practice to use a wrench for a hammer or to use a hammer on a wrench. It *is* good practice

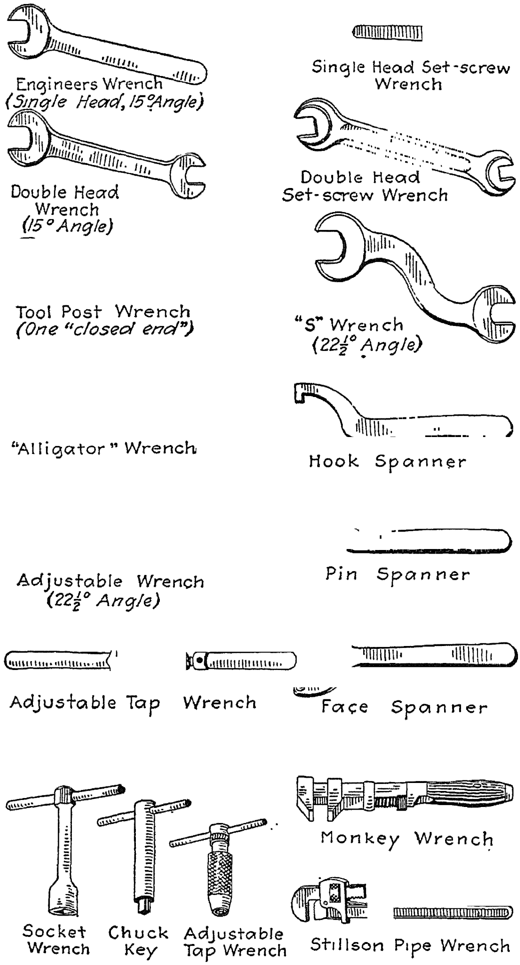


FIG. 211.—Several types of wrenches.

to oil the thread of a bolt or nut and occasionally to oil the screw of a monkey wrench.

6. At times, on heavy work, when one is sure the wrench and also the part to be turned will stand the strain, it is allowable to extend the leverage and increase the mechanical advantage by putting a suitable piece of pipe over the wrench handle. Care and judgment must be used, however, or the wrench will be sprung and possibly the bolt will be twisted off, or the tap or the reamer, as the case may be, will be broken.

7. Oftentimes bolt heads or nuts are so placed in a machine that a movement of the wrench through 90 deg. (for a square nut) or through 60 deg. (for a hexagonal nut) is impossible owing to obstructions. This difficulty is overcome by using for the square nut a 22½-deg. angle wrench and for a hexagonal nut a 15-deg. angle wrench. By turning the wrench over each time it is applied the nut may be turned completely around when the swing of the handle is limited to substantially half that required for a straight wrench. This is illustrated in Fig. 212.

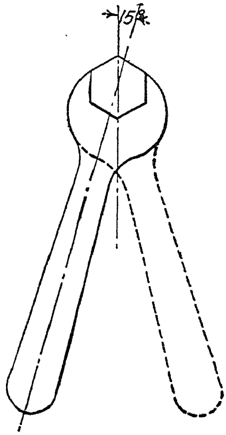


FIG. 212.

8. The ratchet wrench (Fig. 213) is especially useful when only a short swing of the handle is permissible; in fact the multiple ratchet wrench may be used with a swing of only

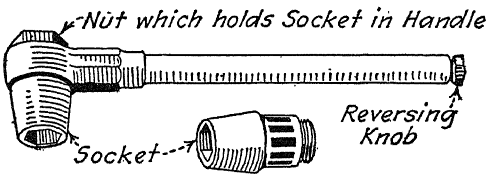


FIG. 213.—Ratchet wrench.

10 deg. An added advantage of the ratchet wrench lies in the fact that it is not necessary to remove it until the bolt or nut is tight.

239. Action of Check Nut.—The principle underlying the proper action of a check nut should be understood. The func-

tion of the check nut is to make more certain that the holding nut will not loosen, that it will stay where it is put. The check nut and the holding nut should be arranged so they bear on opposite sides of the screw thread and thus produce the effect of an extremely tight nut. It is not enough to screw the one nut down against the other. In order to make the two nuts bear on opposite sides of the thread it is necessary to use a wrench on each nut and to turn the first nut back a little as the second is turned down. If the first nut is a fairly close fit on the screw only a small fraction of a turn is necessary. It will be understood from the foregoing that the check nut is the first to be put on, and the second is the holding nut since it bears against the side of the thread that takes the reaction of the part being held. Therefore, if two nuts of unequal thickness are used the thicker one should be put on last in order to obtain the value of its greater strength.

Questions on the Use of Wrenches

1. What is the purpose of the 15-deg. offset wrench? Of the 22½-deg. offset wrench?

2. Explain the use of each of the following wrenches: single-end wrench, double-end wrench, closed-end wrench, spanner wrench, socket wrench, ratchet wrench, pipe wrench, monkey wrench, chuck key, hollow setscrew key, tap wrench.

3. What is the value of a socket wrench?

4. What is a lever? Is a wrench a lever?

5. Why are wrench handles made so short?

6. Should a 12-in. monkey wrench be used on a ⅜-in. cap screw? Give reasons. What do you mean by "sensitive"?

7. If too much force is applied to a wrench, what is liable to happen?

8. What are the two distinct disadvantages of a monkey wrench in the hands of an inexperienced person?

9. Which way should a monkey wrench be applied? Why?

10. What causes the corners of bolts and nuts to become rounded? What does it indicate on the part of the workman?

11. Is a push or a blow more effective in loosening a bolt or a nut? Why?

12. Explain the principle of the action of a check nut.

13. Explain the action of a setscrew.

14. Why is a check nut sometimes used in conjunction with a setscrew?

240. The Hack Saw.—The hack saw is a metal-cutting saw. It is one of the most useful and probably one of the least understood and least appreciated and therefore most abused tools in the shop. The blades are thin and narrow and vary in length by inches from 6 to 16 in. (some makers will furnish them up to 40 in. long). Those used for hand sawing are usually about ½ in. wide, 0.025 in. thick, and 8, 10, or 12 in. long, as desired. The power saw blades are usually somewhat wider (¾ in.) and a trifle thicker (0.032 in.).

Hack-saw blades are made from a high grade of steel scientifically hardened and tempered. They must be very hard and are consequently very brittle. Some manufacturers harden the teeth only, leaving the back soft (flexible-back blades) which renders them less liable to break.

While it seems, and really is, simple and easy to use a hack saw, this is all the more reason why one should be unwilling to exhibit his ignorance by doing this job poorly, either by hand or in the power saw. Learn how a hack-saw blade should be selected and how it should be adjusted in the frame. Be able to judge when it is cutting properly and when it should be discarded. Form the habits of arranging the work close to the vise to avoid spring, and of using judgment when tightening it in the vise.

241. Proper Number of Teeth.—If only a few teeth are broken the saw is ruined. One of the chief causes of breakage of saw blades is due to teeth unsuited to the work. Most shops buy "hack saws" without specifying the number of teeth per inch or the work on which the saws are to be used, and the dealer furnishes a saw with a medium number of teeth (18 or 20 teeth per inch). This saw is all right for solid pieces of steel and cast iron, but is not right for cutting soft materials or tubing. Low-carbon (machine) steel bars as small as 1 in. in diameter may be efficiently cut in a power saw with a heavy blade of 14 teeth per inch; if, however, it is attempted to cut pipe or thin stock or small bars with such a saw the teeth will catch and break. The following saws are recommended by manufacturers for the purposes noted:

Power blades for cutting solids in soft steel—14 teeth per inch.

Hand blades for cutting solids in soft steel—16 teeth per inch.

For general use in hand frames—18 teeth per inch.

High-carbon steel (tool steel) and cast iron—20 teeth per inch

Tubing, brass, copper, drill rod—24 teeth per inch.

Thin sheet metal and thin tubing—32 teeth per inch.

242. Hack-saw Frames are made in various patterns, either in fixed lengths or adjustable, to take 8, 10 or 12-in. blades.

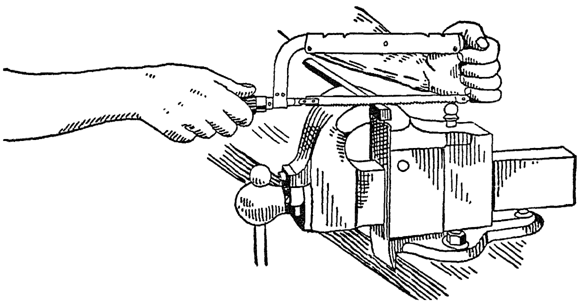


Fig. 214.—Hack sawing.

The blade is fastened in the frame to cut on the forward stroke and should be given considerable tension when in use; taut enough so it cannot buckle and thus be liable to bend or break under the pressure of the stroke. On the other hand, do not strain the blade so much that canting the frame ever so little will break it.

It is easier to cut down than it is to cut sideways or up, and the usual manner of holding the frame (and the blade) is illustrated in Fig. 214. Sometimes, however, it is practicable to hold the frame flat, as, for example, when a long strip is to be cut from a sheet of metal. In such a case, the clips which hold the blade may be given a quarter turn thus setting the

blade at right angles to its normal position in the frame. With the blade so arranged a strip of any desired length may be cut, provided it is no wider than the distance from the blade to the back of the frame.

243. Special Blades.—Hack-saw blades of four different thicknesses (about 0.049, 0.065, 0.083 and 0.109 in., see Fig. 215) are made by several manufacturers for slotting screws and similar work. Such a blade is very handy indeed when a few screws for a special job are needed in a hurry since it saves the time of setting up a machine even if one is available.

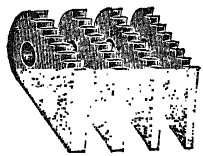


FIG. 215.—Screw slotting hack-saw

244. Hints on Hack Sawing.

1. Hold the work fairly close to the vise to avoid spring and chatter.

2. The smaller pieces will break off easily when the saw cut is two-thirds through.

3. Hold the work securely, otherwise it may loosen under the pressure of the cut and the blade will be broken.

4. Do not, however, pinch a frail piece too hard. Judgment must be used.

5. The tendency is to cut too fast with a hack saw. Fifty or sixty strokes per minute is right for average work. The forward stroke is the cutting stroke, pressure should be relieved on the return stroke.

6. The amount of pressure necessary depends on the kind of material, the width of the cut, and the condition of the blade. If, for example, a fairly thick piece of machine steel is to be cut, considerable pressure will be necessary to make the teeth "bite"; if the same pressure is applied on a narrow piece, or on soft material such as copper, the teeth biting too deeply will catch and probably break. The same reasoning applies when starting a cut on a corner, or on a small rod, or any thin section. Another case of using judgment.

7. Be careful as the finish of the cut is approached or the teeth will dig in the thin section and break.

8. If a saw blade breaks when the cut is only partly finished, start the new blade in another place on the bar. This is especially important when using a power saw. The "set" of the teeth of an old blade is slightly worn and the cut is narrower than the new blade, consequently the new blade will bind if it is attempted to continue this cut.

9. Keep the saw cut straight; when the cut runs, the blade is cramped and will probably break. If it starts to run, give the bar about a quarter turn and begin a new cut; the first cut will help to keep the second one straight.

10. Never use oil as a hack-saw lubricant. A lubricant is unnecessary when hand sawing. In high-speed power sawing where the friction of the blade on the work will tend to heat the blade enough to spoil the temper, water is used to keep it cool. In such a machine provision is made for water to drip on the saw.

11. It is good practice, for the beginner at any rate, to make a small nick with the edge of a file to start the saw cut.

12. Fairly thin sheet metal may be neatly sawed if clamped between two pieces of board. Saw the boards and metal together; this will serve to steady the blade and keep it from digging so easily. If the teeth are too coarse, a blade somewhat worn is better for sawing thin metal than a new saw.

245. Power Sawing.—The general information above applies to power sawing as well as to hand sawing. In addition the following suggestions are offered:

1. Cut the pieces at least $\frac{1}{16}$ in. longer than length given on the drawing for bars up to 2 in. in diameter; allow more for larger pieces.

2. Lower the saw carefully to start the cut.

3. Set a stop for the length if several pieces are to be cut.

4. Production is much greater and more of the saw is used if two bars are cut at the same time.

5. When sawing comparatively thin pieces do not hold them with the edge up, because the pressure of the saw against the thin section will very likely cause the teeth to dig and break.

6. If it seems advisable to saw a pipe or tube with the regular blade, saw a piece of bar stock or a piece of wood at the same time.

7. A blade quite dull will not cut straight down. Put in a new one.

Questions on Hack Saws

1. If you are to cut out a strip say 6 in. long and 1 in. wide, from a sheet of brass, how will you hold the blade in the frame? Why?

2. When using a hack saw, which stroke is the cutting stroke? Which way should the saw be placed in the frame? Does the saw cut on the return stroke? Give reason.

3. What is a fair cutting speed for a hack saw?

4. When are blades with 16 teeth to the inch used? With 20 teeth to the inch? With 32 teeth to the inch?

5. What is the effect if one or two teeth are broken out of a hack-saw blade? Why?

6. How tightly should a hack-saw blade be strung in a frame? Why?

7. Why will a hack saw break if the frame is tipped sideways? If it is not pushed straight? If pushed too hard?

8. When cutting off a piece of flat stock say $\frac{1}{4}$ by 1 in., how should it be held in the vise? Why?

9. Why is a hack-saw blade harder than a wood-saw blade?

10. What makes the hack-saw blade more brittle than a wood-saw?

11. State three common faults in the use of hack saws any one of which may result in breaking the blade.

CHAPTER XIII

LAYING OUT

246. Laying Out is the shop term used to include the marking or "scribing" of center points, circles, arcs, or straight lines

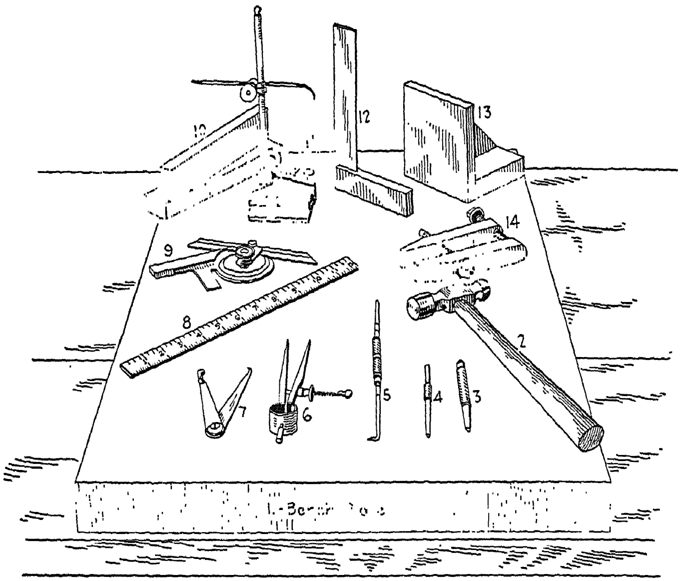


FIG. 216.—Tools used in laying out.

upon metal surfaces, either curved or flat, for the guidance of the workman. It is much used in drill-press work and in shaper and planer work. The layout to be worth anything must be right for the job at hand, therefore the dimensions on the blueprint must be carefully followed and the layout lines made sharp and distinct. The degree of accuracy necessary depends on the job; a great amount of layout work is done on rough castings and it is not to be expected that the dimensions will be in thousandths of an inch. On the other

hand it often requires more calculation and judgment—head work—to lay out a casting so that it will machine to size (especially if it is scant here or there) than it does to lay out *accurately* two or three holes on a finished surface.

247. Tools Used for Laying-out Work (Fig. 216).

1. *Bench Plate,* or *Surface Plate.*—The cast-iron plate of convenient size, ribbed to give strength and staying qualities, machined on top and bottom, and usually in sides and ends. Used when laying out work as a base upon which to rest the work, gauges and other tools. (A surface plate may be differentiated from a bench plate in that its top surface is scraped flat. It is an expensive tool, especially in the larger sizes, and is used for testing work, gauges, etc.) Laying-out plates (floor work) are often 4 or 5 ft. wide and 8 or 10 ft. long.

2. *Hammer.*

3. *Center Punch.*

4. *Prick Punch.*—Similar to center punch except point is much sharper, angle being about 30 deg. while the center punch is ground about 90 deg.

5. *Scriber.*—A slender piece of tool steel 8 or 10 in. long, sharp pointed both ends, one end bent to approximately a right angle, points hardened and tempered, used for marking or scribing on metal.

6. *Divider.*—A tool with hardened-steel points used for scribing circles or laying off distances. It is adjustable and is classified as to size by the maximum opening between the points. Divider points must be slender and sharp.

To set the divider place one point in a convenient graduation line on a scale (for example the 1-in. line) and adjust the other point until it exactly splits the graduation line the correct distance away. Since the graduation lines are V shaped a divider may often be set more nearly exact by feeling than by seeing. Adjust it until no "give" can be felt either side of the V. Check with a magnifying glass if desirable. When the divider is fully opened the exact adjustment is difficult because of the wide angle, consequently a larger divider will be better, or possibly trammels (Fig. 218).

The vernier caliper (Fig. 263, page 378) may be used for accurate settings of dividers or trammels. Near the zero mark on the beam is a tiny indentation in the center of a small circle, and on the sliding jaw is another small cone-shaped mark. Set the vernier to the dimension required and then

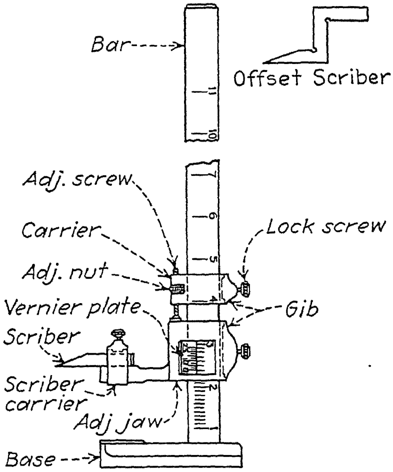


FIG. 217.—Height gauge. (*Courtesy of The L. S. Starrett Company*).

adjust the points of the dividers or trammels, by "feel" or with the aid of an eyeglass, exactly in the two marks.

7. *Hermaphrodite Caliper* (*morphy*).
8. *Scale.*
9. *Bevel Protractor.*
10. *Parallels.*
11. *Surface Gauge.*
12. *Square.*
13. *Angle Plate.*
14. *Parallel Clamp.*
15. *Height Gauge* (Fig. 217).—Used for obtaining the height of projections from a plane surface and for locating and scribing dimensional lines. The upright bar is graduated to read, by means of a vernier[1] scale on the movable jaw, to thousandths

[1] For directions for reading vernier see page 376.

of an inch. The fixed jaw forms a base. The sharp point of the extension of the movable jaw may be used to scribe lines on surfaces that have had the scale removed.

16. *Trammel Points* (Fig. 218).—Practically a beam compass or divider used for drawing arcs or circles having a large radius. An advantage lies in the fact that the point holders are perpendicular to the surface being scribed. It will be noticed in the illustration that caliper legs, a pencil point, or a pen point may be substituted for the divider points. Also a ball is

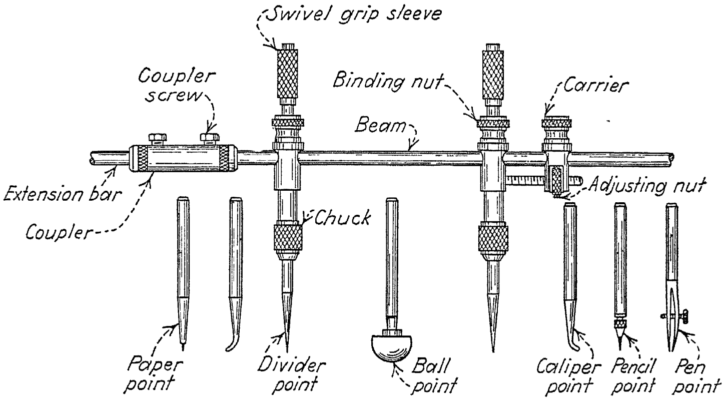


FIG. 218.—Trammels or Universal Dividers. (*Courtesy of The L. S. Starrett Company.*)

provided to enable an arc or circle to be scribed using a hole as a center.

248. Scribing the Lines.—The process of laying out work calls for intelligent reasoning on the part of the workman. It is impossible to give but few rules or directions since each job is practically a problem in itself. Laying out is line work. The surface to be machined is indicated by a line either straight or curved, and the centers of holes are located by the intersections of lines. A double line, or a blurred line, is worse than useless; have the point of the scribing tool *sharp* and *draw one line.* Chalk well rubbed into the surface of a casting will help to make the lines more distinct. White lead mixed with turpentine is quicker and better for the larger work. To

make the lines more distinct on a finished surface apply blue vitriol solution, sometimes called "copper solution," with a piece of waste. To make blue vitriol solution dissolve a small handful of copper sulphate (blue vitriol crystals) in a half pint of water and add a few drops of nitric acid. A cream jar or a pickle bottle having a good-sized neck makes a good container. The diemaker frequently heats the steel for the die or punch until it is blue. In this oxidized surface the finest line is clearly visible.

249. The Operation of Laying Out.—It is usually advisable to work from a given surface or "seat," and if several lines are to be scribed a *base line*, to which the other lines may be referred, should be drawn. The base line may be valuable later to relevel or square up by if it is necessary to move the piece during the layout. If a bench plate is used as a seat for the work and the tools, the surface gauge may be adjusted to a scale held vertically. By carefully adjusting the point of the surface-gauge scriber to the dimensions given on the drawing, parallel lines of the required distances apart may be easily scribed on the work. If greater accuracy is required a height gauge may be used.

The intersections on these lines may then be laid off by scale measurement from a finished surface, from a previously located square or angle plate, or by means of a divider from a given point. Angular lines may be laid off by using a bevel protractor.

If it is not convenient to seat the work directly on the bench plate it may be supported on parallels placed on the plate or it may be clamped to an angle plate and the lines scribed as suggested above.

In drill-press work the centers of the holes are located by the intersections of lines. A light indentation is made at the point of intersection with a prick punch, and using this mark as a center, a circle the size of the hole required is scribed with a divider. After the circles are scribed, make a deeper indentation with the center punch to make it easier to start the drill central.

In layout work if the intersection of lines comes in an opening, as for example in a cored hole or between projections of a casting, it is customary to bridge across the opening with soft wood. A small piece of sheet copper or tin with corners bent down to drive into the wood is used as a surface upon which to scribe the lines and make the slight indentation for the divider point.

Oftentimes in cored work, the core may move a little in the mold which will cause the hole to be out of center. Such a casting may often be saved by compensating on some other surface for the error. In such a case a preliminary layout may be advisable to determine if this surface will clean.

If there is a likelihood of the lines on the rough surface of a casting becoming obliterated, it is good practice to make a series of light center punch marks ¼ in. or so apart along the line. This is not done on finished surfaces.

It will be well for the beginner to check his layout to be sure it is right.

Oftentimes it is necessary to lay off a number of equally spaced holes in a given circumference. The following table of chords (distances between centers of holes) for a circle having a diameter of 1 in. may be used by multiplying the constant for the required number of spaces by the diameter of the given circle.

Number of divisions in circle	3	4	5	6	7	8	9	10	11	12
Length of chord..... Dia. of circle 1 in....	.866	.707	.588	.500	.434	.383	.343	.309	.282	.259

NOTE.—For laying out work see also Part II, Chapter II, Drills and Drilling, and Chapter VII, Planer Work.

Questions on Laying Out

1. What is chalk used for in layout work? When is "whiting" used?

2. What solution is used on surfaces which have been machined, to make the lines show more distinctly? How is it made?

3. What is the difference between a prick punch and a center punch?

4. What do you mean by a light indentation with a prick punch? For what purpose is a light indentation used?

5. Why is a divider set by feel more accurate than when set by sight?

6. What tool is used for a heavy indentation? Why? When is a heavy indentation made? Give reasons.

7. Why is a series of prick punch marks sometimes made to show the location of the circle? When are they used in other line work?

8. On finished work is this necessary? Is it advisable? Give reasons.

9. How are lines parallel to each other or parallel to a base usually scribed?

10. How are lines at an angle to a given line or base usually scribed?

11. How are intersections accurately laid off from a finished surface? From an established point?

12. What do you mean by checking the layout? When is it advisable?

13. Explain the use of the following tools used in layout work: scale, scriber, hermaphrodite caliper, divider, surface gauge, prick punch, center punch.

14. Explain the use of the following tools used in layout work: bench plate, angle plate, parallels, parallel clamps, C clamps, square, bevel protractor, height gauge.

15. What is the distance across the flats of the largest square that can be filed on the end of a cylinder 1 in. in diameter? 2 in. in diameter?

16. To what dimension should a divider be set to space equally ten holes in a 5-in. circle? Five holes in a 10-in. circle?

CHAPTER XIV

CHIPPING, FILING, SCRAPING

250. Chipping.—There is a considerable satisfaction in being able to hold a cold chisel and strike it with a hammer in such a manner as to produce a surface that indicates real workmanship. Almost anyone thinks he knows how to use a hammer, and yet, excepting the screw driver, there is probably no tool in the shop that is more ignorantly or carelessly used.

The shaper, the planer, or the milling machine may usually be regarded as more efficient than the hammer and chisel for the removal of metal, but there are many times when the hammer and chisel are invaluable. There never has been a real machine shop without a cold chisel or a real machinist who did not know how to grind and use one.

251. Cold Chisels.—Cold chisels are made in several shapes (see Fig. 219). The flat chisel is used for chipping flat surfaces, and often for cutting thin sheet metal. The cape chisel is forged to give the greatest strength possible with a narrow cutting edge. It is used for chipping grooves and often holes or slots where a sturdy narrow chisel is needed. The diamond-point chisel is used for cutting V-shaped grooves or for chipping in square corners. The round-nose chisel and the "gouge chisel" are used for such work as roughing out small convex surfaces or filleted corners and for chipping oil grooves, etc. The gouge chisel is used also for drawing a drill to center.

Chisels are made of octagonal-shaped carbon steel (chisel steel) and are classified as to size according to the cross section of the steel. They are carefully forged and should be annealed before being hardened and tempered. Annealing serves to improve the grain of the steel and makes the chisel

stronger. The temper of a cold chisel is drawn somewhat lower than a lathe tool or a drill because it must withstand more of a shock. Only the cutting end is hardened, and this end only half or three-quarters of an inch back from the edge. The proper temper color is purple merging into blue (about 530°–540°F.)

The cutting angle of a cold chisel is about 70 deg., and in flat chisels and cape chisels this angle is included symmetrically

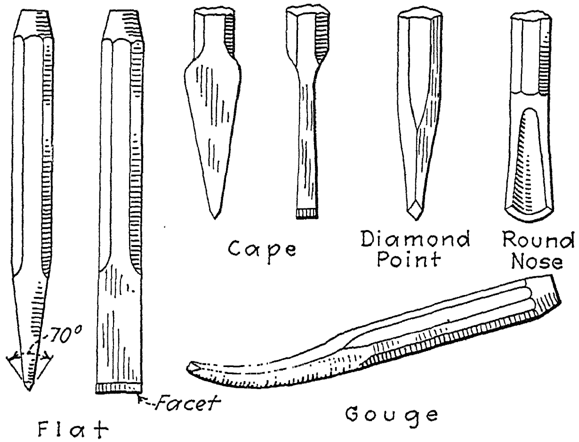


FIG. 219.—Forms of cold chisels.

between two "bevels" or "facets." Back of the cutting edge the facets should be straight, but across the width of the chisel they may be a trifle curved to give the slightly convex cutting edge which many machinists prefer. The other chisels have only one facet.[1]

252. Grinding a Cold Chisel.—Much better control of the chisel is obtained if it is held as shown in Fig. 220 with the left hand resting on the tool rest. Do not hold the chisel too hard against the wheel or the temper will be lost. A flatter and better facet will result if the chisel is held slightly canted and moved slowly back and forth across the face of the wheel, especially if the wheel is at all worn. As the tendency is to

[1] Making a cold chisel, paragraph 315, page 364.

tip the chisel and thus grind more off one end of the facet, the beginner should frequently examine the facet in order to correct this fault. Efficient chipping cannot be accomplished with a chisel on which the facet curves from the cutting edge back; grind it straight. Most beginners will grind the chisel with too sharp a cutting angle; remember that 70 deg. is nearer a right angle than it is half a right angle.

To avoid unnecessary waste of the chisel, practice grinding should be done on an inexpensive piece, say flat stock of cold-

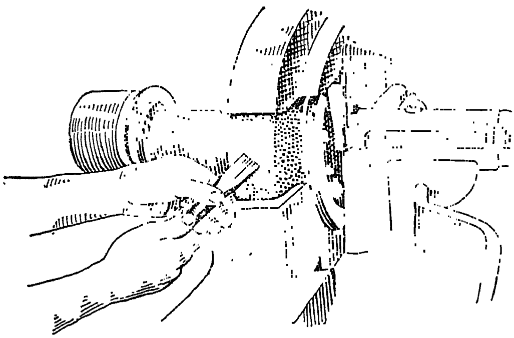


FIG. 220.—Proper way to hold a cold chisel when grinding.

rolled steel 1/8 by 3/4 in., until the skill of grinding equal and parallel facets is acquired.

253. The Operation of Chipping.—The first direction is to avoid gripping the chisel or hammer too tightly. The boy on the New England farm acquired the art of holding a plow after his hands became so tired he could no longer grip the plow handles, and the same principle applies to chipping. Grasp the hammer handle well back toward the end; don't "choke the hammer." Swing the hammer with an easy forearm movement vertically over the shoulder, but hit the chisel with a solid snappy blow. Depending upon conditions, and often upon choice, the chisel may be held with the hand over or under as desired. The surface of the facet bearing on the work is the guide and should be kept parallel to the surface desired. If the chisel is held too high it will "dig," if

held too low the cut will "run away from the line." With close attention to the line and to the chisel edge (*never look at the head of the chisel when chipping*) one will very soon automatically raise or lower the chisel as needed, and the knack of chipping is acquired.

254. Hints on Chipping.

1. It is a good idea to wear goggles, especially if another person is chipping near by.

2. Do not permit too large a "mushroom" to form on the head of the chisel; grind it off occasionally.

3. Do not try to chip with a dull chisel. When necessary to sharpen, grind off only a trifle.

4. When holding work in a vise put a packing block of wood or metal under it to keep it from slipping down in the vise.

5. Use protecting pieces of brass or copper or similar soft material between the work and the vise jaws.

6. Do not hammer the vise handle.

7. Use a light hammer for a small chisel and a heavier hammer (about 1 lb.) for ordinary chipping with a $\frac{3}{4}$-in. chisel.

8. Look at the guiding lines on the surface to be chipped, never at the head of the chisel or a sore thumb will result.

9. Always chip toward the solid vise jaw, if possible.

10. When chipping wrought iron or copper, occasionally lubricate the chisel with oil or soapy water.

11. When chipping cast metal it is best to begin at the ends and chip toward the middle because the force of the blow against the corner is liable to break the corner off below the finish line. And also as a second cut approaches the first cut it is good practice to ease up on the force of the blow in order not to break out a chunk of metal.

12. Don't pet the work—chip it! Count 1001—1002—1003 and so on for time and rhythm.

13. Instead of keeping the edge of the chisel constantly against the chip, most mechanics prefer occasionally to draw it back $\frac{1}{8}$ in. or so, say every two or three blows. This eases

the hand, gives better control and a better cut. Of course the chisel edge is again in place before the next blow falls.

14. Do not take too deep a cut, $\frac{1}{16}$ or $\frac{3}{32}$ in. is enough. The chisel is less liable to break, the cutting edge will stand up longer, and more metal will be removed in a given time. Leave at least $\frac{1}{32}$ in. for the finishing cut to give the chisel a chance to "bite." When finishing be sure the cutting edge is sharp, ease up on the force of the blows but keep them snappy.

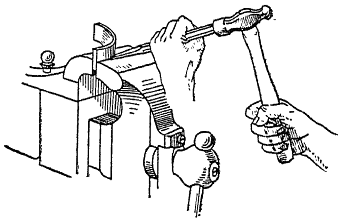


FIG. 221.—Shearing thin material with vise jaw and cold chisel.

15. If the surface to be chipped is fairly wide, use a cape chisel first to cut shallow grooves, and then the flat chisel to remove the stock between the grooves.

16. To cut off a heavy rivet head or similar projection, cut a slot through the middle with a cape chisel and remove the rest with a flat chisel.

17. To cut a hole in a thin sheet use a cape chisel and cut fairly close to the line.

18. To cut off a strip (1) use the vise jaw and the chisel for a shearing cut (Fig. 221) or (2) nick both sides along the top of the vise jaw and then break off the strip.

Questions on Chipping

1. Examine the cutting edge of a properly ground cold chisel. Is the cutting edge straight? Are the facets ground flat?

2. Test the hardness of the chisel near the cutting edge with a file. Test it $\frac{3}{4}$ in. or so back from the cutting edge. Which part is harder? Is it as hard as the file?

3. How far back is a cold chisel hardened?

4. What is the proper temper color for a cold chisel? How does the temper of a cold chisel differ from the temper of a lathe tool?

5. If the cutting edge of a chisel is heated until it is red hot, or even blue, has the temper been destroyed?

6. Why will not grinding off the color restore the temper?

7. After a chisel has been sharpened several times, is it still as hard as ever on the cutting edge? Give reason.

8. How is the chisel held against the grinding wheel? Why not rest the chisel on the tool rest?

9. When the face of the grinding wheel is grooved, how may the facets of the chisel be ground flat?

10. Why is a wet grinder better for grinding tools than a dry grinder?

11. What is the best cutting angle for a cold chisel? Why not 90 deg.? Why not 50 deg.?

12. When grinding a cold chisel, or any other cutting tool, what precaution must be taken regarding the temper?

13. There is always a tendency for the beginner to grind one end of the facet wider than the other end. How do you account for this? How do you correct it?

14. If no grinder is available, how may a cold chisel be sharpened?

15. How dull should a cold chisel become before it is proper to sharpen it?

16. What is a flat chisel? What is a cape chisel? What is a gouge chisel?

17. About what weight of hammer should be used for chipping?

18. Where and how should the hammer handle be grasped? Why?

19. What is the proper position at the vise for chipping? Should the workman raise the hammer over his shoulder or toward his side?

20. How should a chisel be grasped? How tightly?

21. Should the workman look at the cutting edge or the head of the chisel when chipping? Why?

22. At about what angle with the surface being chipped should the chisel be held? Why not a greater angle? Why not a less angle?

23. How do you prevent marring the work when holding it tightly in the vise?

24. How may the piece being chipped be kept from working down in the vise?

25. What kind of a chisel is used for chipping a keyway?

26. Why is it best, if possible, to drill a hole at the end of the keyway to be chipped?

27. Is it proper to lubricate a cold chisel when chipping steel?

28. Why is it advisable to anneal a forged tool before hardening and tempering it?

FILING

255. The Use of Files.—To know the kind of file to use, what to say when asking for the particular file, how to take care of files, and how to use them skillfully should prove a source of satisfaction to almost any boy or man. Filing is one of the accomplishments, like sharpening and using a drill, using taps and dies, soldering, hardening and tempering, etc.,

that are necessary in a machine shop and often valuable in many other places.

When fitting machine parts together there are occasions when a slight reduction in size is required, and the use of a machine tool is impracticable. In such cases the file is most useful. Further, in many classes of work such as diemaking, experimental work, and model work, surfaces must be finished and parts fashioned by filing. Therefore, it is important that a machinist shall be able to use a file skillfully.

Nothing looks much worse in the estimation of a mechanic than a poor job of filing. The art of filing, especially at the

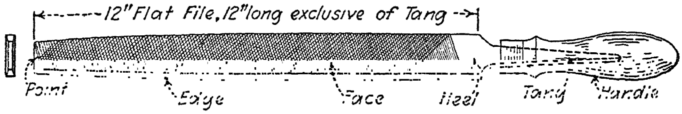


FIG. 222.

bench, is an acquired "knack." The beginner should learn all he can by reading, by observation, and by asking questions; then he may practice intelligently, and in no other line of machine-shop work is it more true that practice makes for skill.

256. Machine-shop Files.—A file (Fig. 222) is a piece of high-carbon crucible steel having teeth cut upon its body by parallel rows of chisel cuts. These cuts are made more or less diagonally across the face depending on the material on which the file is to be used and whether for roughing or finishing. When a file has a single series of cuts across its face, it is known as "single cut." The "double-cut" file has two courses of cuts crossing each other (see Fig. 223). The terms "rough," "coarse," "bastard," "second cut," "smooth," and "dead smooth" refer to the distance apart of the parallel cuts on the larger files (10 in. or over, see Fig. 224), and on smaller files the Nos. 00, 0, 1, 2, 3, 4, 5, 6, 7, 8, refer to the same thing, No. 00 being the coarsest. These terms are relative and depend on the length of the file; a 16-in. "second-cut" file is much coarser than a 10-in. "second cut" and a No. 00 8-in. file is

coarser than a No. 00 4-in. file. The length of a file is always measured exclusive of the tang.

In general it may be stated that a 10- or 12-in. bastard file is used for rough filing at the bench, the second cut for bring-

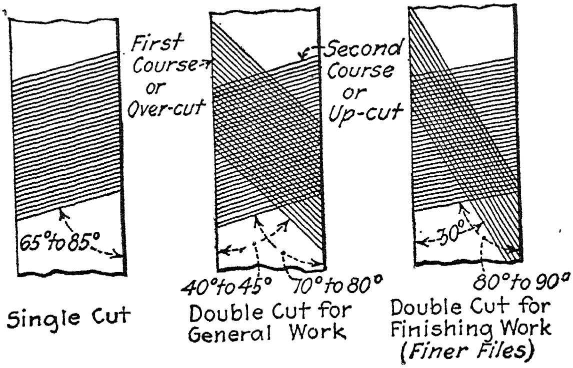


Fig. 223.

ing the work fairly close to a finish, and as fine a file as desired for the finish. The "rough" and "coarse" files and the "dead-smooth" files are not much used in machine-shop work, and of the smaller files the Nos. 00 and 2 are used much more than the finer cuts.

Files may be obtained in almost any desired shape or length and are commonly known either by their cross section as "square," "round," "three square" (triangular), "half round," etc.; by their general shape as "flat," "hand," "pillar," etc.; or by their particular use as "mill file," "warding file," etc. Files are used in all of the metal-working trades; the shapes most commonly used in machine work are illustrated in Fig. 225 and a brief description follows.

The Mill File.—Nearly all of the files used in machine shops are double cut, the most notable exception being the mill file. The mill file is single cut, of substantially the shape of the flat file (Fig. 222) and most commonly 10 or 12 in. in length. It may be obtained with flat or rounded edges or one flat and one round as desired. Bastard or second cut are mostly

used. The chisel teeth (single cut) give a smoother finish than the pointed teeth (double cut) but do not remove the metal so fast. It is usually regarded as better than the double-cut file for lathe work. It is much used for drawfiling (paragraph

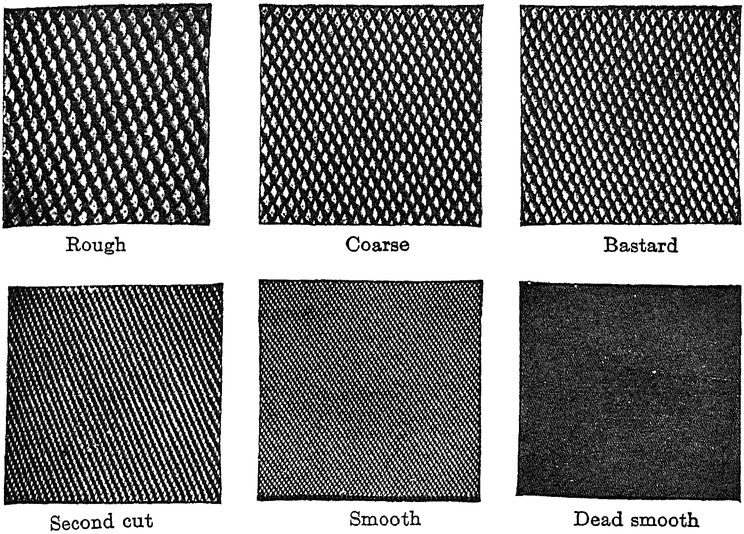


FIG. 224.—In the larger files the relative coarseness of the given cut is known by name, as above; in the smaller files the cuts are distinguished by numbers. Do not get the term "second cut" confused with the term "double cut" as illustrated in Fig. 223.

266), and in the bastard cut is fairly efficient for filing brass or bronze. The mill file derives its name by reason of its extensive use in woodworking mills for sharpening saws and planer knives. In machine-shop work it is often called "float" file or "lathe" file, but "mill" file is the correct name.

The flat file is rectangular in shape and tapers slightly narrower and thinner toward the point and toward the heel (Fig. 222). It is the most commonly used file in the shop for general work. Usually 12 in. long and bastard cut but may be obtained in lengths from 6 to 16 in. in any cut.

The hand file is parallel in width with faces slightly convex. The hand second-cut file is an excellent file for removing feed

marks and for bringing flat surfaces fairly close to the finish. The hand smooth is the favorite finishing file for flat surfaces and is often used for finishing surfaces of round work revolving in a lathe.

The pillar file is similar to the hand file except that it is narrower. An 8-in. pillar file is light and "handy." It is adaptable for a great variety of filing operations and is one of the most popular files in the shop. Used usually in No. 00, 2, or 4 cut.

The square file is used for filing the smaller square or rectangular holes, for finishing the bottoms of narrow slots, etc.

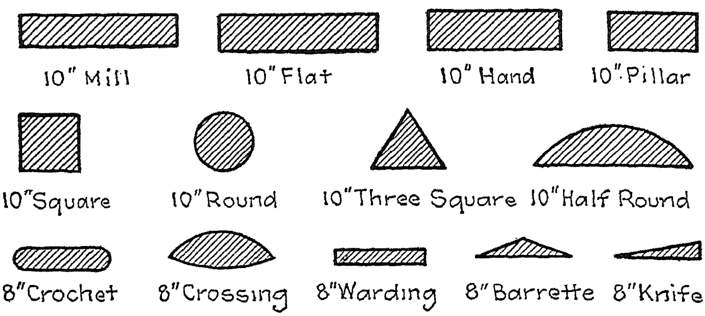


Fig. 225.—Cross sections of several files.

The round file is used for enlarging round holes and for finishing round corners. It is generally tapered and the small sizes are often termed "rat-tail" files.

The three-square files are double cut on all three sides and the edges are very sharp. These files are especially valuable for finishing surfaces that meet at less than a right angle, and for backing off special taps, counterbores, etc., that frequently must be "home made."

The half-round file is one of the most useful files in general machine-shop work. It may be obtained in any length and cut.

The crochet file has both edges rounded. It is useful when filing against a filleted shoulder or a rounded corner of a hole.

The crossing file, sometimes called the "shadbelly," is often used in place of the half round. Each side of the file has a different curve which feature frequently is of great convenience.

The warding file is a very thin file. It is essentially a locksmith's file for making the ward notches in keys. It is however very useful in the machine shop for filing slots and notches, and for finishing the sides of narrow grooves.

The barrette file has a flat triangular shape with teeth on the wide face only (safe back). It is a most useful file for finishing the sharp corners of many sorts of slots and grooves.

The knife file is used instead of the barrette file in similar work where the thinner cross section and the safe back of the latter are not necessary.

The files mentioned above are only a few of the shapes manufactured. By referring to a catalogue of files it will be observed that there is manufactured a size, shape, and cut of file for practically every purpose and any material. When a machinist or toolmaker asks for a file he gives the *length*, the *name*, and the *cut;* for example, "10-in. hand second cut" or "8-in. pillar No. 2."

257. The Safe Edge.—The mill file and the flat file have single-cut teeth on both edges; the hand file usually has teeth on one edge only, the other edge being termed the "safe" edge; the pillar file has two safe edges. If a safe edge is desired on any file it is easy to grind off the teeth. As a matter of fact a sharper corner may be obtained with a file so ground.

258. Convexity of Files.—Most files are made with the faces slightly convex lengthwise or "bellied." There are good reasons for this. If when filing a broad surface all the teeth were in contact, it would require too much pressure downward to make the file "bite" as well as forward to make it cut; this would mean practically double work and also make it more difficult to control the file. If the face of the file were straight, to produce a flat surface every part of the stroke would have to be perfectly straight. This is impossible. If a file were cut with flat faces and warped ever so little in

hardening (and this is impossible to avoid) then one side would be concave and useless for flat work.

259. Taper of Files.—The convexity of a file should not be confused with the taper of a file. A flat file has faces which are convex, and it also tapers slightly in width. Certain files, for example the square, round, and triangular files, are more adaptable for a variety of work if they taper from near the middle to a very small cross section at the point, and are generally so made. There are occasions, however, when it is preferable to use a file of uniform cross section; such files are termed "blunt."

260. File Handles.—One important thing concerning files is too often disregarded; the file should be provided with a suitable handle properly fitted. The size of the handle depends on the size of the file, and the nature of the job. On the larger files the handle should be of a size that may be easily grasped; if too large or too small it will tire the hand in heavy filing. On the smaller files the handle should be of a size that will give balance to the file. When using a 4- or 5-in. file a piece of leather belting cut to a convenient shape makes an excellent handle. A wooden handle is fitted as follows: Drill a hole in the handle, of a size equal to the average thickness of the tang, and to a depth about equal to the length of the tang. Heat the tang of an old file to a dull red and force it about two-thirds its length into the hole and quickly withdraw it. Plunge the handle in water to stop the burning and then drive it on the new file, being careful not to split it. Practically the whole of the tang should be fitted to the handle.

261. Care of Files: Pinning.—There is always a tendency, especially when filing narrow surfaces or corners, to have the file "pin," that is, small particles of the material being filed get wedged in front of the teeth of the file and scratch the work. Keep the file clean either with a "file card" (*a*, Fig. 226), or by pushing the dirt from between the teeth with a piece of soft steel, brass, or copper flatted thin on the end (*b*, Fig. 226).

Pinning is caused often by bearing too hard on the file, especially on the finer cut files. The worst kind of pinning is caused by hard usage of a new file. The new file should be used with great care until the small burrs on the ends of the teeth are worn away. Files are expensive cutting tools and

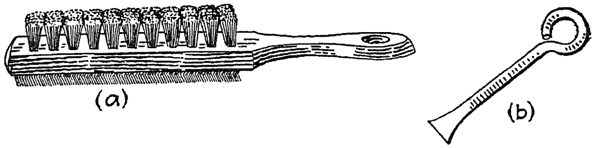


FIG. 226.—File cleaner.

it is a sure sign of ignorance or carelessness to throw them in a drawer or on the bench. Be orderly and careful; it pays.

262. Cross Filing.—Pushing the file endways, under more or less pressure against the work, is called cross filing, or merely "filing." No pressure is applied to the file on the return stroke; it should not be removed from the work but may rub lightly.

263. Holding the File.—Most filing is done by holding the file in both hands. The file should be held in one hand only in

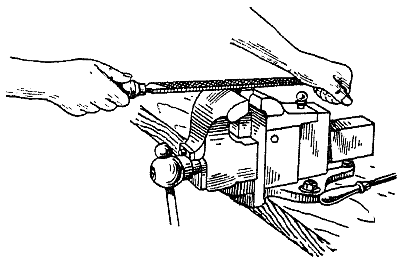


FIG. 227.

especially delicate work where the suitable file is too small to be held in both hands. The proper way to hold the file for heavy filing is shown in Fig. 227. Grasp the file handle in the right hand, with the palm of the hand against the end of the handle and the thumb on top. Cover the other end of the file with the base of the thumb of the left hand and curl the fingers under.

For the lighter finishing cuts the position of the right hand remains the same, but the left hand may be changed to the position shown in Fig. 228. This gives better control of the file.

The beginner will usually grasp the file too firmly. He will generally acquire about the proper grip after his hands get tired and cramped.

264. Position of the Body When Filing.—Skillful filing is a *knack* and to acquire this knack it is essential that conditions

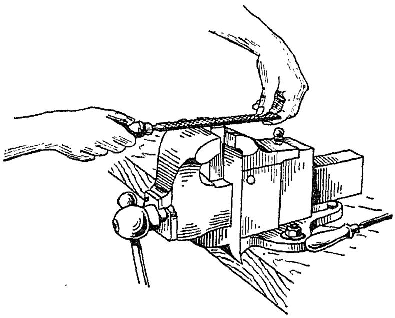


FIG. 228.

must be right. It is easier to do a thing well by the right method than it is by the wrong method, but having once learned by the wrong method it is difficult to acquire the right. The height of the work to be filed in the vise should not be above the level of the workman's elbow as he stands erect, therefore the shorter boys should be provided with platforms to stand on. Filing at the bench, especially the heavier filing, calls for a certain harmonious action of the arms, body, and legs. Stand with the left foot pointing toward the bench, the hollow of the right foot 8 to 12 in. from the left heel, and bend the body slightly forward at the hips. Hold the file as shown in Fig. 227 with the right arm bent to about 90 deg. and the left arm somewhat nearer straight. Lean forward slowly for about two-thirds of the stroke, bending the knees slightly, and at the same time, push with the arms. During the last third of the stroke, keep pushing with the arms but bring the body back slightly to nearly the original position. Then bring

back the file lightly on the work to position for another stroke. Keep the file level or the work will be rounded instead of flat. The great fault is too much speed. Bear on hard but take slow strokes.

265. Operation of Filing.—If the work is cast iron the scale should be removed, possibly by chipping, before an attempt is made to file the surface. A few strokes on cast-iron scale will ruin a file. When rough filing it will be found that "crossing the stroke" (Fig. 229) at short intervals will rest the arms. It also serves to show the beginner where he tends to file the hardest, thus helping him to keep the surface straight, and practice in keeping the surface flat and straight is practice in learning to file. The work should be tested occasionally with a scale or with the blade of a square (Fig. 230). Test it crosswise, lengthwise, and diagonally. A common fault is to rock the file. Either an over-arm or an under-arm rocking action will produce a convex surface; push the file as straight as possible.

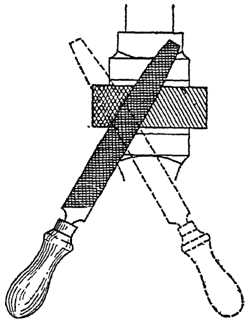


Fig. 229.

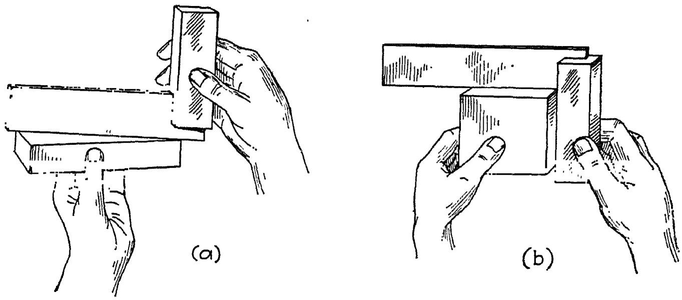


Fig. 230.—(*a*) Testing for flat. (*b*) Testing for square.

If the surface being filed is to finish square with another surface, care must be taken to *keep it fairly square when roughing* because if one corner is filed $\frac{1}{16}$ in. low, it means either that the work is spoiled or that the sixteenth thickness must be filed off the whole surface. Test it frequently with a

steel square as shown in *b*, Fig. 230. When finishing the surface the defects will show more plainly if the work and the square are held between the light and the eye. Remember that a machinist's steel square is a tool that will not stand rough handling. If it is the least bit "out" it is worthless.

Another advantage of the convexity of the file will by this time have become apparent. The skillful workman can control the file to make the few teeth that touch the work at one time cut just about where he desires. If the middle, or one corner, or one edge is a little high he files off the high spot and this is the art of filing a true surface.

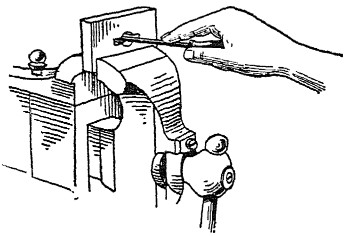


FIG. 231.

Filing is a skill, to acquire which one needs to pay careful attention and to have patience; "all of a sudden" you have it and filing is easy. If one has attained a reasonable amount of proficiency in two-hand filing—the sense of file balance and control—he will find no particular trouble in learning to file with one hand. A stool of a height that will bring the workman's shoulder to about the level of the work should be provided. The file may be grasped in several ways; the position of the hand as illustrated in Fig. 231 is suggested as one giving ease in operation and good control.

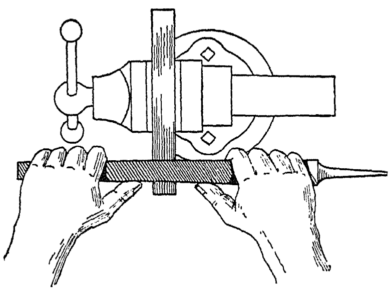


FIG. 232.—Drawfiling.

266. Drawfiling.—When it is desired to "line" or grain a piece of work (either flat or round) lengthwise it may be done by drawfiling. Hold the file as shown in Fig. 232. Keep it flat, and bear on as hard as necessary both directions of the stroke, being careful not to scratch the work with a dirty or

dull file. Removing the handle in certain cases may give better balance of the file. The single-cut file is usually regarded as best for drawfiling. After the piece is filed it may be polished by moving an emery stick, or a piece of emery cloth under the file, back and forth in the same manner as drawfiling.

267. Filing Soft Metals.—Filing brass, solder, lead, etc., with the ordinary double-cut file is very unsatisfactory for the reason that the teeth quickly become clogged with chips which are difficult to remove. The *brass file* (Disston) has been designed to overcome this difficulty and also to produce a better finish with a coarser file. It is made with deep teeth and open bottoms; the up cut is on a longer angle than usual and the over cut is almost straight across. If such a file is not available a single-cut file is usually more satisfactory than a regular double-cut file.

The *curved-cut file* (vixen), Fig. 233, has proved very efficient for filing soft metals. The shape of the grooves between the

Fig. 233.

teeth reduces to a considerable extent the clogging of the teeth when filing aluminum, brass, copper, etc. The curved-cut file with its chisel edges gives an excellent finish when filing round work in a lathe. Further, it has very free cutting action which makes it desirable when considerable stock, iron or steel as well as the softer metals, is to be removed. It is especially economical for the reason that unlike other files it may be advantageously recut or sharpened.

268. Needle-handle Files.—The very smallest files are made in the form of "needle-handle files" (Fig. 234). They are much used in fine die work and are also very useful in delicate finishing touches in a variety of filing jobs. They are made in 4- to 6-in. lengths, only a third to a half of which is file shaped and cut, the remainder forming a slender handle.

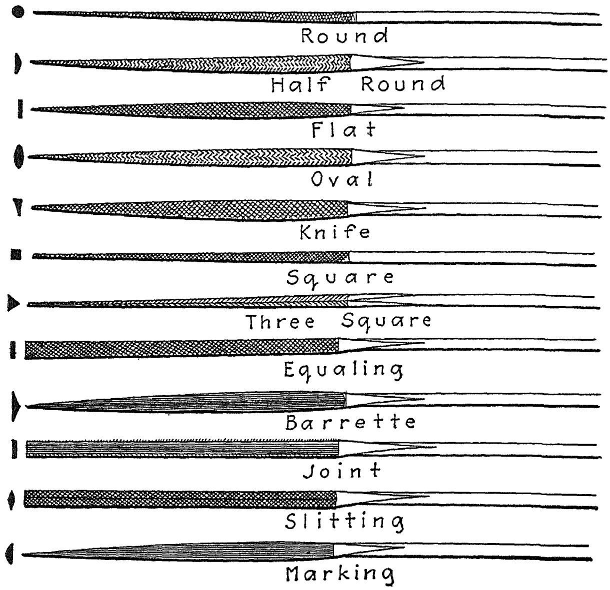


FIG. 234.—Needle-handle files.

Don'ts in Filing

Don't use a file without a handle (except when drawfiling).
Don't use a loosely fitting handle.
Don't use a worn-out file.
Don't use a dirty file.
Don't use a new file on narrow edges.
Don't use a good file on cast-iron scale.
Don't use a fine file for filing soft metal.
Don't use a bastard file for finishing.
Don't use a smooth file for roughing.
Don't let the file hit the vise jaws.
Don't allow files to scrape together.
Don't put your fingers on the cast-iron surface being filed.
Don't push the file too fast.
Don't file too much before testing the work.

Questions on Files and Filing

1. What is one reason for having the file slightly convex or "bellied"?

2. If the file were not bellied, and warped in hardening, would its usefulness be impaired? Explain.

3. If the file were not bellied, would it be easier to "take hold" or harder? Why?

4. What is the effect when filing if the right hand tends to go down and the left hand rises slightly?

5. What is the effect when filing if the right hand tends to rise and the left to go down?

6. Is it easy to file the edges and produce a convex surface? Give reason.

7. It may be stated that in order to produce a flat surface with a 10- or a 12-in. file, a harmonic movement of the arms, body, and legs is necessary. What does this mean?

8. What do you mean by a "knack"? Is filing a flat surface a knack? How is a knack acquired?

9. What is meant by crossing the cut in filing?

10. Should the file be lifted from the work on the return stroke? What is the reason?

11. What are the differences between a flat file and a hand file?

12. What is the difference between a bastard file and a second-cut file?

13. In what way does the mill file differ from the other files?

14. What do you understand by bastard? Second cut? Smooth?

15. How is the length of a file measured?

16. What is the difference between a "double-cut" file and a "single-cut" file?

17. What is the difference between a "double-cut" file and a "second-cut" file?

18. Which is the easier metal to cut with a file, cast iron or wrought iron?

19. How should the scale on castings be removed before filing the surfaces?

20. What is the reason a file should not be used to remove the scale from cast iron?

21. When should the coarser files be used? When should the finer files be used?

22. On narrow work, should an old file or a new file be used? Why?

23. What can be done to keep cast-iron filings from clogging the file?

24. How is the handle properly fitted on the tang of a file?

25. What is meant by a "safe edge" on a file? When is it advisable to use a file with a safe edge? If necessary could you grind a safe edge on a file?

26. What commonly used file has two safe edges?

27. Is a half-round file half round?

28. On what kind of surfaces is a half-round file used? What is the purpose of having teeth cut on the flat side?

29. How is the cut of a file designated in the smaller sizes?

30. What is a file card? How may a piece of brass or copper rod be made into a most efficient file cleaner?

31. What causes "pinning"? How may the "pin" be removed?

32. In your judgment, why should a single-cut file be best for filing in a lathe?

33. What is a needle-handle file?

SCRAPING

Scraping in machine work is the operation of *pushing* off, rather than *peeling* off, a small chip with a very sharp tool

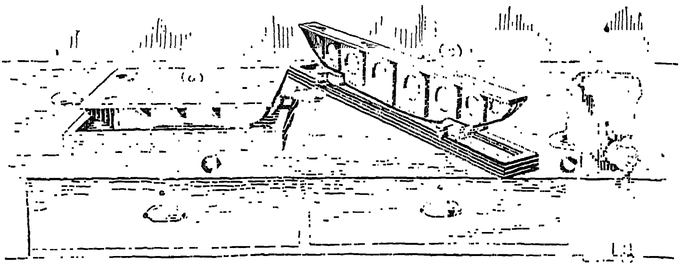


FIG. 235.

having no rake. For example, hand reaming is really a *scraping* action. Ordinarily, however, the term is confined to the finishing of flat and curved surfaces with hand "scrapers" as here explained.

269. Reasons for Scraping.—(1) It is practically impossible to produce a true flat surface in a machine or with a file. (2) Most of the flat bearing surfaces are iron surfaces, and the condition of an iron surface as it comes from a planer or shaper or milling machine is not suitable for a first-class bearing surface; first, because it is not exactly flat and true, and second, because even the sharpest cutting tool does not produce the *close grained* and *smooth* surface that is necessary for such a bearing. (3) There are many curved bearing surfaces, for

example in the bearing boxes and caps for shafts, spindles, etc. These surfaces, whether of cast iron, babbitt, or bronze, must be scraped to obtain the alignment and fit necessary in high-grade work.

270. Tools Used for Scraping a Flat Surface.—To produce an accurate flat bearing surface the "high spots," although they are only two or three thousandths of an inch high, must be located and scraped off. The tools used for locating the high spots and for otherwise gauging the shape and accuracy are special gauge plates as for the ways of a lathe or a dovetail slide; surface plates (*a*, Fig. 235), which are made in a variety of shapes and sizes; and iron straight edges (*b*, Fig. 235), which are practically long narrow surface plates. These plates are scientifically designed to retain their shape. They are very expensive and should be handled and used with the greatest care.

The tools used to remove the high spots are called scrapers. Fig. 236 shows the commonly used forms of scrapers for flat work either of which may be made of a size convenient for the job at hand. The flat scraper (*a*, Fig. 236) for general machine-shop use is usually about the size of a 10- or 12-in. hand file. It is drawn down on the end to about $\frac{1}{16}$ in. thick, hardened, and the "snap" taken out, that is, heated just enough so that a drop of water will bubble on it. In other words, a scraper should be as hard as it is possible to make it. Scrapers are often made from old files but they are not so good as those made of special scraper steel. The hook scraper (*b*, Fig. 236) is used for flowering or frosting, which are the terms used for the more or less regular scroll or patchwork design which is sometimes used to "finish" a scraped surface.

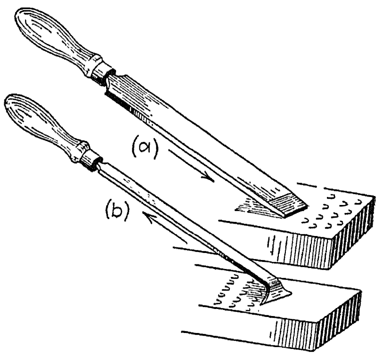


FIG. 236.

They are also used for scraping surfaces where it is inconvenient to use a flat scraper, as for instance in the angle of a dovetail bearing surface.

271. Sharpening the Flat Scraper.—The taper sides of the cutting edge are ground flat and smooth, and the end is ground square edgewise but slightly convex lengthwise. The object of the slight curve of the cutting edge is to enable the user to cant it ever so little to one side or the other to scrape exactly the spot desired without danger of scratching under the surface of the work with the corner of the scraper. After grinding, the scraper is oilstoned, the flat sides first and then the end. When stoning the end hold as shown in Fig. 237 and move the scraper for a distance of about 3 in. back and forth on the oilstone. Tip the handle ever so little forward and bear down on the push stroke, easing up on the return. When one cutting edge is sharp, turn the scraper half around and oilstone the other edge. Do not hold the edge square to the direction of the stroke but about 45 deg. as shown in the figure. This will tend to give a better edge if the oilstone is uneven, and also give the slight convexity desired.

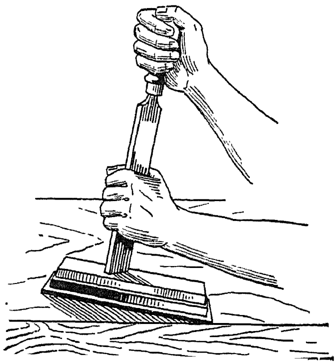


Fig. 237.

272. The Scraping Operation.—To mark the high spots a thin application of Venetian Red or Prussian Blue paint is rubbed on the surface plate, and the work rubbed on the plate or the plate on the work, whichever is the more convenient. The high spots on the work will then show plainly, being marked with the paint. There will show the first time only a few spots and these will be more or less isolated. These are scraped off and the operation of marking repeated. As the scraping continues, the number of the spots will increase, and

strange as it may seem, the greater the number of spots evenly distributed the more perfect the scraped surface.

Figure 238 shows the correct position for holding the scraper. With the flat scraper the forward stroke is the cutting stroke

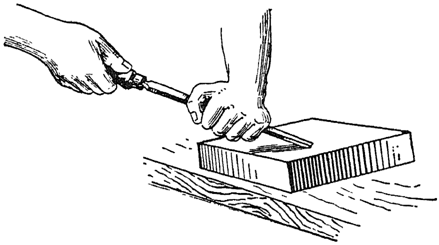


FIG. 238.

and the usual stroke is seldom over ½ in. in length. The effect should be clean cut and smooth—no scratches. Keep the scraper *sharp;* when one edge becomes dull turn it over and use the other edge. Oilstone it when necessary and grind it a little every three or four times it is oilstoned. Not more than two or three thousandths should be scraped; if the work is more uneven it probably should be machined again although it may perhaps be more economical to file it. As the spots increase in number and decrease in size, considerable judgment must be used as to just where and how much to scrape. Turpentine may be used instead of the paint when close to the finish to show the high spots which will appear bright. The turpentine acts also as a lubricant between the work and the plate, which is often an advantage, and further it serves to make the scraper cut better.

273. Hints on Scraping.

1. Do not allow any oil, not even your fingers which are naturally oily, to touch the surface being scraped.

2. Until the surface is substantially true it is necessary, to accomplish anything, to scrape hard. As the spots begin to show evenly over the surface ease up on the chip.

3. Dipping the scraper occasionally in turpentine (or water) will help it to cut easier and better.

4. When roughing, especially, try to keep the cuts about square in shape and cross them in succeeding courses. This will help to make the marking more easily distinguishable.

5. The best way to apply the paint is with a rag swab or by hand. Apply it more generously when roughing than when finishing.

6. Be extremely careful that no grit or dirt gets on the surface plate and that none remains on the work when being tested. Keep the paint box covered.

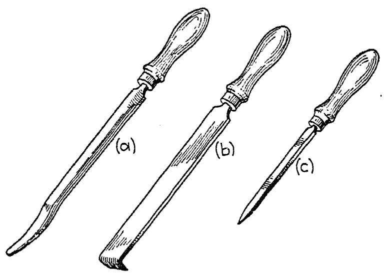


FIG. 239.

7. Use the whole plate—not one spot.

8. Keep the scraper *sharp* or it will scratch. Scratches spoil a scraped surface.

274. Scraping Curved Surfaces.—Round bearings and other curved surfaces have often to be scraped to fit running or sliding parts. The scrapers used are shown in Fig. 239. The half-round bent scraper *a* is mostly used. It has two cutting edges and the cutting stroke may be either toward or away from the user as desired. The proper way to hold this scraper is shown in Fig. 240. For the larger curves the scraper shown at *b*, Fig. 239, is preferred by many; it is ground to conform to the curve of the work and sharpened to cut on the pull stroke. Scraping a curved surface to fit a shaft or a gauge is not particularly different in principle or operation from scraping a plane surface.

The three-cornered scraper (*c*, Fig. 239) is used to "break" (remove the sharp edge), or round as desired, the corners of

holes or other curved edges. It is usually made from an old file by grinding off the teeth. The three corners of the body

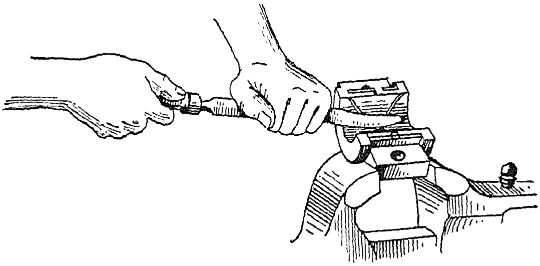


FIG. 240.

back of the cutting edges should be well rounded in order not to cut the hands of the user.

Questions on Scraping

1. Is there a difference between the cutting angle of a flat scraper and a scraper for curved work?

2. Is the end of a flat scraper straight? What shape is it lengthwise? Crosswise?

3. What is the difference between a scraper for a round bearing and a three-cornered scraper? Could a three-cornered scraper be used for scraping a round bearing?

4. Why is a scraper for round bearings curved slightly?

5. When is a three-cornered scraper used? How is it usually made in the shop?

6. Of what kind of steel is a scraper made? Why?

7. How is a scraper hardened? How is it tempered?

8. What care must be taken when grinding a scraper?

9. Why is a scraper oilstoned after grinding?

10. How is a scraper for a round bearing oilstoned?

11. How is a flat scraper held when being oilstoned?

12. What are the advantages of a scraped surface?

13. What is a surface plate? Why must great care be taken of a surface plate?

14. How is a flat bearing surface tested?

15. How is a round bearing surface tested?

16. Why is a thin coating of red lead or Prussian Blue applied to a surface plate? Which is better? Why?

17. How do you determine when a surface is sufficiently scraped?

FORGE WORK

CHAPTER XV

SOLDERING, BRAZING, AND BABBITTING

275. Soldering is the operation of joining metal surfaces by means of a surface fusion to these surfaces of another metal (solder) of a lower melting point. Common soft solder is a compound or an alloy of equal parts of tin and lead and melts at about 430°F.

276. The Principle of Soldering.—Metals have an affinity or attraction for each other, that is, under certain conditions they will combine forming a new substance called an alloy. Many conditions govern the alloying of metals. One of these conditions, usually but not always, is heat. Mercury will combine with gold, silver, copper, and zinc at ordinary temperatures. This can be observed by rubbing mercury on a piece of copper. On the other hand lead and copper and tin do not combine at ordinary temperatures, but they do combine readily at higher temperatures.

Soldering is a process of forming an alloy of the various metals present on the surfaces of the pieces to be soldered together, and involves the fusing into a common mass of all the metals present at the point of connection. This fusion of metals takes place at a temperature below the melting point of some of them. As stated above soft solder is an alloy of tin and lead in equal parts; tin melts at a temperature of 450°F. and lead melts at a temperature of 618°F.; nevertheless, if a piece of lead be placed in molten tin the two will quickly combine. If equal amounts of lead and tin are thus combined the melting point of the alloy formed will be about 430°F. which is lower than that of either metal.

When two pieces of metal are soldered tightly together they are held together because of the fact that a very thin layer of alloy (an alloy of the solder and the metal) has been formed on the surface of each piece through the application of heat and they have fused together. If it is not a tight joint the solder between the surfaces will fill the space as well as hold the pieces together. The underlying principle of soldering is that the comparatively low temperature of the soldering copper will serve to melt the solder and form an alloy on the surface of the materials being soldered, and fuse these alloys together, or to the solder between them, as the case may be.

277. Cleaning the Surfaces.—One of the essentials of the soldered joint is that there be contact between absolutely clean surfaces. The preliminary cleaning may be done with a scraper, file, or piece of emery cloth. However, a metal surface that has been thus cleaned immediately oxidizes through exposure to the air. That is, the oxygen in the air combines with the metal and forms a thin coating on the surface of the metal. The layer of oxide, no matter how thin, will not form an alloy with another metal; soldering cannot be accomplished between surfaces which are oxidized. Scale, grease, and dirt must be removed, and further, the oxides of the metals must be removed before a union of the metals can take place. Since oxidation cannot be entirely avoided the layer of oxide must be removed at the instant of soldering. This is accomplished by the use of fluxes.

278. Fluxes.—The action of the flux as it vaporizes is to carry off with it the oxide. There are several kinds of fluxes used in soldering. The flux most commonly used in machine-shop work for soldering brass, galvanized iron, and steel is "killed" muriatic (hydrochloric) acid. Muriatic acid is killed by adding small scraps of zinc a little at a time until the acid has eaten all it will and ceases to boil. As the acid eats the zinc, hydrogen in the acid is liberated. The killed or cut acid is no longer muriatic acid, but is known as chloride of zinc. While it is being killed it should be moved away from any machines or tools that may be rusted by the escaping

fumes. For soldering brass, copper, tin, and steel it is inadvisable to use a flux which is too strong. It should be diluted with about 25 per cent water.

There are available, in any hardware store, certain patented fluxes that are suitable when soldering copper, galvanized iron, and polished steel. They are usually in the form of paste and may be obtained in various sizes of containers.

Muriatic acid, or raw acid as it is called in soldering work, is used as a flux when soldering pieces to cast iron or black sheet iron or zinc. When raw acid is used for a flux the work

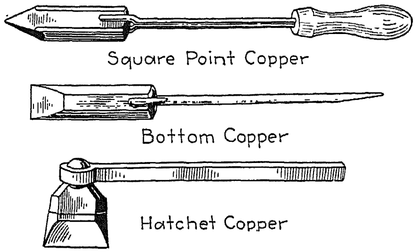


Fig. 241.

should be rinsed in water, after soldering, to stop the action of the acid.

For soldering tin or copper powdered rosin makes the best flux. It is sprinkled along the joint and when melted has a tendency to flow into the seam. Borax is used as a flux when using hard solder (spelter), that is, when brazing (see paragraph 283). The borax acts similarly to rosin in that it works its way into the seam or joint by capillary action, and the solder follows.

279. Soldering Coppers.—Soldering coppers (Fig. 241) are made in different shapes and of varying weights. The square-pointed copper is used more for general work than such shapes as the blunt roofing copper or the chisel-shaped bottom copper. A small copper should not be used on heavy work because it will not keep its heat long enough. On the other hand a copper that is too heavy is unnecessarily clumsy for light work. It is very necessary that the copper be properly "tinned" and

that in heating care be exercised not to overheat. If it is overheated a scale of copper oxide is formed on the point and this scale being practically a nonconductor of heat renders the copper almost useless until it is retinned.

280. Tinning a Copper.—To tin a soldering copper it is necessary to file or otherwise smooth and clean the end for about ¾ in. back. The copper is then heated enough to melt solder, after which the flux and the solder are applied. There are several ways of applying the flux and solder when tinning the copper, the method employed depending almost altogether on the material at hand.

1. The soldering iron may be dipped in the acid and then the solder applied.

2. The point of the copper may be rubbed with a piece of sal-ammoniac and then the solder applied. This is the quickest and best way.

3. A piece of solder and some powdered rosin may be placed on a brick and the heated copper rubbed thereon until it is tinned.

281. Dipping Solution.—If the copper while being heated becomes discolored by reason of the kind of fuel used (charcoal, gas, or gasoline) it may be cleaned by dipping it quickly in a solution made by dissolving a teaspoonful of powdered sal-ammoniac in a quart of water.

282. Soldering Operation.—When soldering it is often convenient to "tack" one piece to the other, that is, a few drops of solder are put here and there to hold the piece in position. When finishing the operation care must be taken to let one portion cool before proceeding to the next. The copper should be so applied that as much of the available heat as possible may be utilized and further it must be placed in such a position on the work over the joint that the solder will flow into the seam and not merely along the outer edge.

Oftentimes it is good practice to solder two or more thin pieces together and machine them as one, afterwards melting them apart. It is usually best when soldering steel to steel to be sure both are tinned, because steel does not alloy with

solder as readily as does copper or brass. If the steel pieces are properly cleaned and heated, flux applied generously, and solder rubbed on with the heated copper and the excess rubbed off with a piece of waste, the surfaces will show bright with a thin coating of solder. Like the soldering copper they are "tinned." If a little flux is applied and the tinned surfaces are held together and heated they will be perfectly soldered or, as sometimes called, "sweated" together.

BRAZING

283. Brazing.—A much stronger joint can be made by brazing than by soldering. Brazing is a process of joining metal parts which is similar to soldering except that "spelter" is used instead of solder. Spelter is a compound of copper and zinc and is often called hard solder. It is usually about half and half copper and zinc; adding more copper up to two-thirds copper and one-third zinc serves to produce a stronger joint, but makes it more difficult to work. A spelter made of half copper, three-eighths zinc, and one-eighth tin makes an excellent spelter. Spelter is used in either granular or wire form. Brass rod is a combination of substantially two-thirds copper and one-third zinc (with a small amount of lead added to make it machine easier) and if prepared spelter is not available brass filings will make an excellent substitute.

The flux used for brazing is powdered borax. A small amount is mixed with water to form a paste and applied to the surfaces to be brazed before they are heated. During the process of brazing the dry borax is sprinkled on the joint where it melts and flows between the surfaces to be joined. The use of too much flux should be avoided because it hardens the surfaces of the joint and makes the filing and finishing difficult. When the flux begins to flow the spelter is placed on the joint and the heat continued until the spelter flows into the joint and no longer. A spatula for placing the flux and also the spelter on the joint may be made by flattening the end of a steel rod of suitable diameter and length.

To produce a strong joint it is necessary to have the surfaces fitted and held together tightly. Copper, brass, wrought iron, malleable iron, or steel may be brazed. Care must be taken when brazing copper or brass not to overheat and melt the work.

Questions on Soldering and Brazing

1. Of what materials is soft solder composed?
2. Steel is an alloy. Brass is an alloy. What is an alloy? Is solder an alloy?
3. When a piece of solder is melted on a piece of copper is a new alloy formed?
4. What is meant by the term "fuse"?
5. When a piece of solder is melted on a piece of copper, why does the solder cling to the copper?
6. What is rust? What is meant by oxidation?
7. If a piece of steel is polished, it will retain its brightness in ordinary temperature for several days or maybe weeks, but if heated sufficiently it will quickly become almost black. Why is this?
8. Why will not solder cling to an oxidized surface?
9. What is the action of a flux?
10. Why is a flux used in soldering?
11. What flux is commonly used in machine shops? How is it made?
12. How is the soldering copper tinned?
13. What care must be taken when heating a copper? Why?
14. How are pieces of metal sweated together?
15. What is the difference between soldering and brazing?
16. Of what materials is spelter composed?
17. What flux is used for brazing?
18. How is the flux applied to the surfaces to be joined?

BABBITTING

284. Babbitt.—Babbitt metal is an alloy of copper (4 parts), tin (88 parts), and antimony (8 parts). It is widely used as a lining for bearings for the following reasons: The bearings (boxes and caps) do not have to be bored; the shaft or spindle of the machine may be aligned and the babbitt melted and poured around it. It is practically an antifriction metal. It is strong, tough, and durable. It may be readily machined if necessary, and scrapes much easier than cast iron or bronze. When the bearing becomes worn the babbitt is easily broken

out with a chisel, remelted, and poured to form a new bearing surface.

Lead is added in the cheaper grades of babbitt. This is all right for shafts having light duties to perform but to produce good babbitt bearings requires a high grade of Babbitt metal, of which there are several brands in the market.

285. Babbitting a Bearing.—There are several methods of pouring babbitt bearings:

1. The bearing box or housing is cylindrical in shape and the metal is poured around the shaft, or around a rod or "babbitting mandrel" the exact size of the shaft to be used.

2. The metal to form the bearing is poured around a mandrel somewhat smaller than the exact size of the shaft. The bearing is afterward bored to the size required.

3. The bearing is split on a center line horizontally, separating the upper and lower halves. Both the lower and upper parts of this bearing are poured at the same time. Pieces of cardboard called liners are placed between the two, and against the shaft, holes being made in the liners near the shaft for the passage of the metal from the upper to the lower part. (The narrow portions of metal may be easily broken when the cap is removed.)

4. The bearing metal is poured in the upper and lower parts separately, the lower part being poured first, after which the upper part or cap is adjusted and the Babbitt metal poured in to form the lining for the cap.

It should be understood that in either of the above methods (3) or (4), the metal may be poured around a shaft or mandrel to the exact size and merely scraped to form a suitable bearing surface, or it may be poured around a shaft or mandrel or a suitable rod somewhat smaller than the exact size of the bearing required and afterward machined to size. If before machining, light blows are struck with a ball-peen hammer over the entire surface, it will serve to harden the surface and make the grain of the alloy closer when machined and a more durable bearing is produced. This method is considered the best.

The bearing boxes are usually cored to the required size. On the smaller sizes the cored holes are straight and cylindrical. To prevent the babbitt lining from working loose small holes may be drilled at right angles to the axis of the bearing and the metal running into these holes, when poured, serves to tie the lining securely to the box or cap. To serve the same purpose in the larger bearings the box or cap may have two or more dovetail recesses cored parallel to the axis. Where a thrust load is taken on the bearings, the casting is either cored or counterbored somewhat larger in the end taking the thrust in order to provide against any tendency for the load to loosen the lining.

If the shaft or babbitting mandrel is painted with a mixture of graphite and gasoline it will make removal easier. This is especially true in a solid bearing, that is, one in which the metal forms a solid sleeve around the shaft. A coating of lampblack applied by holding a candle flame against the shaft will serve the same purpose.

It is advisable to preheat the mandrel and also the casting of the bearing. Otherwise the cold casting may chill the babbitt enough to prevent its filling the space, and in any event the bearing surface will not be as smooth as if the box and shaft were heated. The heating may be done with a blowtorch or by any convenient means.

If the bearing is to be poured "to size" considerable care should be taken to align the shaft. If many boxes are to be babbitted it will save time to make a suitable fixture for holding the shaft in position.

Fire clay mixed with water to the consistency of putty may be used to close the openings between the shaft and the box and thus keep the metal from running out. It may be well to cut cardboard washers and back them up with the clay.

The babbitt metal should be slowly heated until it will quickly burn a pine stick to a dark brown. Too much heat will injure the metal and when insufficiently heated it does not pour well. If a small amount of crushed rosin is put in

the babbitt just before pouring a smoother bearing surface will result.

Care must be taken that no water comes in contact with the melted babbitt. Even a few drops of water will cause the metal to spatter.

The pouring must be continuous. If it is interrupted the additional metal will not fuse with that already poured and the lining will be cracked. It is sometimes necessary when pouring large bearings to use two ladles, pouring from both at the same time.

Questions on Babbitting

1. Of what materials is babbitt made?

2. What causes the difference in the grades of babbitt?

3. What is babbitt used for?

4. Name three advantages of babbitt for the purpose for which it is used.

5. Explain how the cheapest type of babbitted bearing is produced.

6. Explain the principal features of the best type of babbitted bearing.

7. Would a bearing such as suggested in question 5 be suitable for shafts having heavy duty to perform? Why?

8. Would a bearing such as suggested in question 6 be practicable for a 1-in. diameter shaft? Why?

9. How is a small babbitt bearing kept from working loose?

10. How is a large babbitt bearing kept from working loose?

11. When a thrust load is to be taken on the bearing, what provision is made to keep the babbitt in place?

12. When proceeding to babbitt a bearing, how do you get the boxes ready?

13. How do you get the shaft or mandrel ready?

14. How do you separate the cap from the box? Why?

15. Why do you heat the box and the shaft?

16. How do you prevent the babbitt from sticking to the journal?

17. How do you prevent it from running out at the ends of the box?

18. How much babbitt do you melt? Why?

19. Why is a bearing spoiled if the pouring is interrupted?

20. How slowly should the babbitt be melted? Why?

21. How may you tell when the babbitt is hot enough to pour?

22. What does it matter if the babbitt is too hot?

23. How does a little rosin affect the babbitt?

24. When do you put it in?

25. What will cause the melted babbitt to "spatter"?

CHAPTER XVI

HARDENING AND TEMPERING STEEL

286. Introduction to the Theory.—Without going into detail, a word may be said about what steel is. All steels are alloys of iron, that is, compounds made of iron mostly, but combined with the iron are other substances that give the alloy (steel) its special qualities. It is by knowing the effects of these various substances combined with the iron, that the different kinds of steels are made, selected, and used for the particular purpose. The Ford Motor Company, for example, makes 40 kinds of steel to use in the car itself, and for the tools used in making the car. A trifling amount of an added ingredient may change the nature of the steel.

Some of the alloying elements that are often referred to in machine-shop practice are nickel, manganese, chromium, molybdenum, tungsten, and carbon. Countless experiments and tests have been made and are continually being made to determine the characteristics of combinations of one or more of these elements with iron, in all manner of proportions; and the effects of all kinds and degrees of heat treatments on each product. The one element always combined with iron is *carbon*. Carbon is met with every day in the form of charcoal, coke, graphite, and lampblack. It is the content of about 1 *per cent of carbon* that makes steel capable of being hardened.

The iron itself is mined as *iron ore* which is about half iron. Iron ore is smelted in a blast furnace to produce *pig iron* which is about 90 per cent iron. Pig iron is further refined in huge furnaces to make all kinds of cast irons, wrought irons, and steels. In these furnaces, of various kinds, the iron, in a molten condition, may be brought to practically a pure state, or it may be made to contain the elements desired.

When iron ore is smelted the molten iron absorbs carbon from the fuel (coke), usually 4 or 5 per cent. In further refining the iron to make castings and wrought iron and steel, the amount of carbon is reduced. In *cast iron* it is not very much less, and the iron is brittle. Cast iron cannot be forged.

In *wrought iron* most of the carbon has been eliminated, it is nearly pure iron. It is not brittle like cast iron but is tough and ductile and is readily forged (wrought). Low-carbon steel, also, is almost pure iron. In composition low-carbon steel and wrought iron are nearly identical, but wrought iron, due to the way it is made, has a more fibrous structure.

The first method of using iron ore was, no doubt, to make as pure iron as possible. It was called wrought iron. It could not be hardened and the next step learned was to recarbonize the wrought iron and make a product that would harden. This was called *steel.* Swords are still in existence that were made of this steel and marvelously forged, hardened, and tempered, hundreds of years ago.

The best grades of steel are still made by refining the pig iron of practically all of its impurities, including carbon, and then adding the exact amount of carbon and other elements desired. Although in former times the amount of any element—carbon, for example—that steel contained was judged by experts, today it is the result of an almost exact science, plus skill. The amount of carbon in steel is ordinarily referred to by "*point*"—1 per cent equaling "100 point."

Since the machinist must know how to harden and temper high-carbon steel (tool steel) a few words of explanation of *carbon in iron* should be interesting and helpful to the beginner in machine-shop practice.

Carbon will chemically combine with iron, that is, it will dissolve in the iron and form a new substance, carbide of iron. This carbide of iron is called *cementite.* The chemical formula is Fe_3C [three atoms of iron (ferrite) and one atom of carbon]. Every piece of steel contains its proportion of carbon in the form of cementite, in a matrix of iron. It is in the form of cementite that carbon is made into a *solid solution* with iron to produce steel. In other words, the carbon in steel is, *first, chemically combined* with some of the iron to form molecules of *cementite,* and, *second,* the minute crystals of cementite are intimately mixed in the mass as it cools from the molten state, and form a solid solution with the rest of the iron.

Under the microscope the structure of the steel slowly cooled from a high temperature (red heat) appears to be made of one or at most two of three different particles or crystals. They are ferrite, cementite, and pearlite.

Ferrite is pure iron, named from the Latin word *ferrum.*

Cementite is the iron carbide Fe_3C.

Pearlite is the name given to the *saturated* mixture of ferrite and cementite. It is, in fact, the mixture of cementite in iron that will give 0.90 per cent (nine-tenths of 1 per cent) by *weight* carbon in iron. When the steel has under 0.90 per cent (90 point) carbon the particles are in the form of microscopic layers of *ferrite* and pearlite because there is not enough cementite to saturate the iron. When over 90 point, the particles are in layers of *cementite* and pearlite because there is too much cementite to be perfectly mixed. And when the steel is just 90-point carbon content it is composed entirely of *pearlite* because the proportion of ferrite and cementite is right. Steel with 0.90 per cent carbon, the saturation mixture, is known as a *eutectoid,* under 0.90 per cent as *hypoeutectoid,* and over 0.90 per cent as *hypereutectoid.*

In the short discussion that follows only steel of around 0.90 per cent carbon, the so-called carbon tool steel, or merely "tool steel," will be considered.

When a piece of tool steel is gradually heated, there will suddenly (at about 1350°F.) occur a distinct change in the appearance of the steel—"the shadows disappear." With proper instruments a *drop* in the temperature may be recorded. At this time a very wonderful change has taken place in the steel, the *structure* is different. The temperature at the time of the change is called the *decalescence point.* If heated above the decalescence point and slowly cooled the reverse change takes place, and a slight *rise* in the temperature may be noted. This is called the *recalescence point.*

(In steel having less than 90-point carbon, one, and in the case of low-carbon steel (below 30-point carbon), two similar changes occur before the decalescence point is reached. The temperatures at which they take place are known as the

critical points, and the temperature range between the points is called the *critical range*.)

It is at the decalescence point, or a little above, that tool steel should be quenched to produce, in the hardened steel, the smallest grain size and the maximum hardness.

When tool steel has been heated to the decalescence point or slightly above, the normal structure (pearlite mostly) has been *transformed* to another state, or structure element, which is called *austenite*. This is defined as the solid solution of iron and carbon as it exists above the transformation point (decalescence point), or as it is *set* by rapid cooling, or *preserved* by the presence of a *retarding element* as in 12 per cent manganese steel. It is impossible to set but a small part of the mass of red-hot steel, as it is quenched, in the form of austenite; there are, in fact, several other structures in the piece of hardened steel. The names, besides austenite, that have been given to these definite structures are: *martensite*, which is the principal structure element in very hard steel; *troostite*, which is the transition structure element next to martensite; and *sorbite*, which is softer than troostite. From sorbite the structure of the steel changes, as it cools gradually, to the original pearlite. It will be clear that if the steel is suddenly cooled, or *retarded* by some agent like manganese, and is not allowed to change back slowly to pearlite, then the structure (hardness) will depend upon whether martensite, troostite, or sorbite was preserved.

The above is interesting as containing the theory of tempering steel. When hardened steel is reheated, the changes back towards pearlite that were retarded (set) by quenching are caused to take place. That is, the structure changes from austenite-martensite through the other structures or it may be stopped when at the desired physical condition. This is called tempering the steel. Tempering reduces the brittleness and makes the steel tougher, in proportion to the degree of heat used. Incidentally some of the hardness is sacrificed.

Briefly stated, hardening a piece of steel is the process of heating it thoroughly to or a little above the decalescence

point, quenching it in a suitable bath, on the ascending heat if possible, and thus setting or trapping the steel in the austenitic-martensitic condition or structure. Tempering a piece of hardened steel is the process of reheating the steel to have the combination of martensite-troostite-sorbite which will give the degree of toughness required.

287. **Steel,** then, is an alloy of iron and carbon.[1] The carbon content is usually not over 1.40 per cent (140 point)—high-carbon "razor-temper" steel, and may be as low as 0.10 per cent (10 point)—"dead-soft" steel. The nearer pure iron the steel is the softer it is, the more easily it forges and welds, and the more readily it machines. Consequently 10- to 20-point carbon steel (dead soft) is used for making chains, rivets, etc., and 20 to 35 point, being stronger, yet machining easily, is used for bolts, shafting, etc.

Steel between 30 and 70 point is used for heavy forgings in engine work and for car axles, rails, etc.

Steel over 75-point (0.75 per cent) carbon content may be hardened and tempered.

288. **Machine Steel and Carbon Tool Steel.**—Steel under 50-point carbon does not harden perceptibly, and is called "low-carbon" or "mild" or "machine" steel. Steel over 75 point will harden and is called "high-carbon" or "tool" steel and to differentiate it from the "high-speed" steels, which will presently be referred to, it is often called "carbon tool steel." While the 75-point steel will harden and is suitable for hammers, crowbars, etc., it is not suitable for the cutting tools used in machine shops. Steel for drills, taps, reamers, etc., requires more carbon and the best carbon tool steel for cutting tools has a carbon content of from 90 to 100 point. Steel of substantially 100-point carbon is used for drill rod and spring steel; for punches and dies; for hack saws and files, etc. Steel when heated is changed in its structure and in its properties. Steel of around 100-point

[1] As mentioned on page 333, *alloy steels* have other alloying elements than carbon. Excepting *high-speed steel* (see page 343) they are not discussed in this book.

carbon when heated to a temperature ranging from 1375 to 1400°F. attains the finest structure (grain size) which it can have, and if quenched in cold water the moment it has attained this heat, it is trapped in this perfect structure and in a condition of extreme hardness.

When reheated sufficiently (about 400°F.) a noticeable change begins to take place in the quality of hardness, and when heated some 600°F. the steel has lost all of its brittleness and most of its hardness. That is to say, carbon tool steel changes from glassy hard to practically soft—the whole range of temper takes place—between the temperatures of 400 to 600°.

FIG. 242.

289. The Hardening Heat.—If steel is not heated enough it will not harden; if overheated it is injured; if very much overheated it is ruined. The proper temperature is determined largely by the carbon content, the less carbon the greater heat. For instance, a hammer made of 75-point carbon steel will require 1425 to 1450°F. while a reamer made of 100-point carbon steel requires less heat (1350 to 1400°F.) and will be injured if heated to 1450°F.

Metcalf's Experiment.—To show the effect of different degrees of heat on the strength, appearance, and hardness of the heated and quenched tool steel, take a piece of steel about ½ in. in diameter and 6 or 7 in. long, and with a pointed tool, cut shallow grooves ½ in. or so apart and number the sections (Fig. 242). Put one end of the piece in the fire and allow it to get white hot (very much overheated) before the other end is hot enough to harden at all. Plunge into water and after thoroughly drying break off as many sections as are necessary to show: (1) That the overheated end is weak—breaks easily—and the grain is very coarse; (2) that as succeeding pieces are broken the blow necessary to break them is increasingly greater, and the grain size is finer; (3) that the end

insufficiently heated is not hard—it may be easily filed (use the corner of a worn-out file for testing the hardness); (4) that in the sections along the center portion the greatest hardness and the greatest strength (resistance to the blow) are accompanied by an uneven structure and the finest grain.

Metcalf's experiment may be made more quickly by using a strip of tool steel say ⅛ in. by 1 in. by 5 in. Overheat one end while the other end is occasionally dipped in water to keep it from getting even a low red heat. When the hot end is *white hot* dip the whole piece in water until cold. Reheat carefully just enough to melt solder. Now catch the piece in a vise in order to break it in two *lengthwise* (see Fig. 243). With the upper half projecting above the vise, break off about two-thirds of the length and bend the rest a little. The broken edge should give a complete picture, showing the overheated enlarged grain, the perfectly heated close grain, and the underheated unhardened part that bends and does not break. The overheated end breaks much easier than the properly heated part. Why?

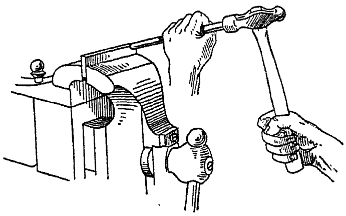


Fig. 243.—Breaking the flat experimental piece. Use any suitable rod or piece of scrap to hammer against and thus avoid tendency to hammer the vise.

At the moment a piece of steel is exactly the right heat for hardening, a change takes place in the structure. It is not difficult for the beginner to gauge this proper heat quite closely if the heating is fairly slow and uniform, because at this moment the darker portions of the mass fade into a uniform red heat. Try heating any small piece of steel, for example the unhardened part of the experimental piece, and notice this change. The corners may appear hot enough (be careful, don't heat too fast, don't let the corners get too hot) while the rest of the piece appears a darker red. Soon, however, the whole piece assumes an even heat and then is the time to quench it. That is, quench it "when the shadows disappear."

290. Hints on Hardening.—1. Do not let the air blast come in contact with the steel more than is necessary because it oxidizes the surface—that is, burns out the carbon and consequently the surface will not harden. If such a surface cannot afterward be ground off—for example in such a tool as a tap—the tool will be worthless.

2. Heat the piece slowly, thoroughly, and evenly, otherwise it is liable, when quenched, to be soft in spots, or warped or possibly cracked.

3. Do not overheat the steel or it will not hold an edge. If by any chance it is overheated, do not hold it until it looks about right and then quench it, this will do no good. It will be much better to allow the steel to cool, then reheat it to the proper temperature and quench. If a piece is much overheated the surface is blistered. Such a surface is permanently ruined.

4. Do not use the fire hotter than necessary, else the corners or thin sections will heat too quickly—before the rest of the piece is hot enough.

5. Do not lay a long slender piece so that it can bend of its own weight when heating, or it will be bent when hardened.

6. Quenching should be done on the ascending heat. Remove the work from the fire to the quenching bath as quickly as possible.

7. Quench the long work lengthwise and the flat work edgewise to avoid warping. If the sections of the work vary greatly in size, quench the large section first to avoid cracking.

8. To avoid soft spots keep the piece moving in the quenching bath, thus keeping it in constant contact with the cold bath and away from the steam produced.

9. Keep the work in the bath until it may be touched without burning the hand, or until the instant the "vibration" in the tongs is no longer felt, then remove it and *reheat until a drop of water will bubble on the surface.* This is called "*taking the snap out.*" The work as it comes from the bath is under enormous hardening strains. Taking the snap out means

reducing these strains to a considerable extent and has saved hundreds of pieces.

10. For the smaller cutting tools or work of a frail cross section or larger pieces that need not be especially hard, a bath of "quenching oil" is most satisfactory. For average work, however, cold running water is the most efficient quenching bath.

11. The hardness of a piece of steel may be tested with a file. Do not ruin a good file, use the corner of an old 8- or 10-in. pillar or half-round file. A file will not "bite" into hardened steel until the temper has been drawn to a certain extent.

291. Tempering Experiment—Temper Colors.—Take a piece of drill rod ¼ in. or more in diameter and 5 or 6 in. long and harden it. Polish it bright and wipe it clean. Hold it in a pair of tongs and pass it lengthwise slowly back and forth over the fire allowing the heat to act most on the farther end and gradually less toward the end in the tongs. Examine occasionally and observe that soon the silver brightness will disappear on the hottest end giving way to a faint-yellow or light-straw color. As the rod is heated a little more the light straw will creep up the rod and be followed successively on the end and along the rod by a dark straw—brown—brown with purple spots—purple—bright blue—dark blue.

Color	Degrees Fahrenheit
Pale yellow	430
Light straw	450
Dark straw	470
Brown	490
Brown with purple spots	510
Purple	530
Bright blue	560
Dark blue	600

These are called temper colors and are very useful many times to indicate the degree to which the steel has been reheated, or as the hardener says, how much the temper has been "drawn." With the corner of an old file test the hard-

ness of various sections of the work. It will be noticed that the end which has been drawn to a blue files fairly easy, and that it is increasingly difficult to make the file bite as the portion of the rod showing light straw is approached.

The temperatures corresponding to the various colors are as shown in the table on page 341.

The colors themselves are merely different thicknesses of films of oxide which are caused by heating the steel to different degrees. The fact that a piece is temper colored does not indicate that it has been hardened; a soft piece will color the same as a hard piece.

292. Hints on Tempering.—1. Machine-shop tools of carbon steel will give good results if tempered about as follows:

Color	Tools
Pale yellow	Cutting tools for lathe, planer, shaper
Light straw	Milling cutters, drills, reamers
Dark straw	Taps and dies
Purple	Center punches, cold chisels
Purple verging into blue	Screw drivers

2. Common soft solder melts at approximately the heat required for tempering a milling cutter, reamer, or similar tool. Heat the piece carefully until it will melt the solder rubbed on it.

3. In tempering, the piece should be heated slowly, carefully, and thoroughly at some little distance from the fire, otherwise the thinner sections will be drawn lower than necessary before the heavier part is drawn enough.

4. The piece being tempered should be moved constantly, to bring each portion, where an even temper is desired, to the same degree of heat. Such a tool as a tap or a reamer or an end mill should be passed slowly back and forth over the flame and turned at the same time. The distance from the fire will depend on the intensity of the heat; do not attempt to draw the temper too quickly.

5. Small pieces may be tempered by moving them about on a heated plate until the temper is sufficiently drawn.

6. The most satisfactory method of tempering is by means of oil (tempering oil or cottonseed oil) heated to the desired temperature. The pieces are allowed to remain in the oil until they are thoroughly tempered. It does not injure them to remain longer than necessary in the oil provided the temperature does not rise.

7. When hardening and tempering cold chisels, screw drivers, or forged lathe, shaper, and planer tools, etc., it is customary to proceed as follows: Heat the piece somewhat farther back than is necessary, harden as much of it as desired, brighten the hardened part with a piece of emery cloth, or a piece of a broken abrasive wheel, and then allow the heat in the unhardened portion to draw the temper. When the proper temper color is observed dip the whole piece. One precaution should be emphasized: when hardening a piece as above, keep it moving up and down in the bath through a distance of half an inch or so, in order not to cause too sharp a line between the hardened and unhardened portions. If the piece is not moved it is liable to crack at the water line. Another favorite method of hardening and tempering a chisel is explained in paragraph 295.

293. High-speed Steel.—In the two experiments made (paragraphs 289 and 291) it was demonstrated (1) that heating a piece of carbon steel above 1400°F. produced a coarse grain, and (2) that the range of temperature for tempers from glassy hard to practically soft was from about 400 to 600°. This is true for carbon steel but not true for high-speed steel.

Mushet, the famous English steel expert, discovered many years ago that a steel containing 1.5 per cent carbon, from 5 to 8 per cent tungsten, and about ½ per cent manganese would harden in an air blast, and would hold its temper until it was practically red hot. Mushet steel, or air-hardening steel as it was sometimes called, was therefore capable of taking a faster and heavier cut since the friction did not burn out the temper and break down the cutting edge. But Mushet steel could not be readily machined and did not come into popular use.

It was not until the American investigators Messrs. Taylor and White discovered the special properties of steel with substantially 0.68 per cent carbon, 18 per cent tungsten and 5 to 6 per cent chromium, and developed a special process of heat treatment of this steel, that the possibilities of high-speed steel were recognized. Since then (1898) dozens of different brands of high-speed steels have been manufactured.

There are two great advantages in the use of high-speed steel for cutting tools. (1) Greater leeway may be given in the heat-treating temperature—it is fairly difficult to spoil a piece of high-speed steel unless it is melted. (2) The temper is not ruined and the cutting edge does not break down even when the tool is red hot, consequently more than double the production can be obtained with a high-speed cutting tool than with a carbon-steel tool.

High-speed steel costs more than carbon steel and for many tools is no better than carbon steel. For milling cutters, lathe tools, and much-used sizes of drills and machine reamers, it is worth much more than the extra cost.

The Taylor-White process is used for the treatment of tungsten-chromium steel.

Taylor-White Process

1. *The high-heat treatment.*
   a. Heat slowly to 1500°F.
   b. Heat rapidly from 1500°F. to a white heat (2200°F.).
   c. Cool rapidly (kerosene bath) to below 1550°F.
   d. And cool either rapidly (kerosene) or slowly (air blast) to the temperature of the air.
2. *The low-heat treatment.*
   e. Heat to a low red (1150°F.) for about 5 min.
   f. Cool either rapidly or slowly to the temperature of the air.

For molybdenum-chromium high-speed steel the treatment is substantially as above except that in *b* it is unnecessary to heat the steel to a white heat, 1850°F. is sufficient.

294. Case Hardening and Pack Hardening.—If low-carbon steel is heated sufficiently when in intimate contact with carbon or a carbonaceous material it absorbs in a certain length of time enough carbon into its surface or "case" to make high-carbon steel of this surface for a depth of several thousandths of an inch. Then if the steel is already red hot, or reheated until it is red hot, and quenched, the *case* is *hardened,* while the interior, being machine steel, remains unhardened and tough as ever.

There are several case-hardening compounds in the market; the best known is cyanide of potassium. The steel is heated in the cyanide or other case-hardening compound in a special pot to a temperature of about 1350°F. for a length of time, 15 min. or more, and then quenched in water.

Be very careful in the use of cyanide of potassium, it is a *deadly poison.* Be careful, too, not to let any drop of water get into the cyanide, from wet tongs or otherwise; it will explode a mass of the cyanide.

Pack hardening serves the same purpose as cyaniding. The pieces are packed in iron boxes with carbon (usually charred bone), sealed with fire clay, and heated for several hours at approximately a hardening heat. This carbonizes the surface. They are allowed to cool in the furnace to normalize them, and afterwards are reheated to about 1350°F. and quenched. Or possibly the smaller pieces being thus treated may be quenched directly from the hot packing box without normalizing.

Questions on Hardening and Tempering Steel

1. What element is always combined with iron in making steel?

2. What is cementite?

3. What is pearlite?

4. What is meant by the decalescence point?

5. What do you understand by the different structures that occur in steel as it cools from the recalescence point?

6. What is meant by martensitic structure or form?

7. Name two somewhat softer structures of heat-treated steel.

8. Explain the process of tempering tool steel.

9. With reference to the structure elements, explain the theory of tempering tool steel.

10. What is the difference between mild steel and carbon tool steel?
11. What is the difference between hardening and tempering?
12. What is the meaning of the term "point" in speaking of steel?
13. Explain how "Metcalf's experiment" is made.
14. Explain the value to the machinist of Metcalf's experiment.
15. In hardening steel why should it be heated thoroughly and evenly?
16. In tempering steel why should it be heated slowly and carefully?
17. What effect does overheating have in hardening?
18. What effect does overheating have in tempering?
19. What is the difference between "burning" a piece of steel and "burning" the temper?
20. What is meant by "taking the snap out" of a piece of steel?
21. What are temper colors? What do they indicate?
22. Does a cutter drawn in oil have temper colors? Give reason.
23. What are some of the advantages of high-speed steel?
24. How is high-speed steel heat treated?

CHAPTER XVII

HAND FORGING IN A MACHINE SHOP

295. Introduction.—It is probably true that a machinist or toolmaker does more hardening and tempering of steel than he does forging; nevertheless, to understand the fundamentals of hand forging, of heating, holding, and hammering a piece of wrought iron or steel saves time. Such knowledge often makes unnecessary a sketch or drawing or an explanation to the smith in another shop.

On succeeding pages are definitions of the common hand-forging operations, introductory descriptions and illustrations of the forge tools, together with a few notes concerning common processes.

296. Forging Operations.—The following are definitions of the forging operations for shaping properly heated wrought iron or steel by means of hammer blows.

1. *Drawing* or *drawing-out* is the process of lengthening a piece of stock while the cross-sectional area is being reduced.

2. *Spreading* is the process of making a part wider and correspondingly thinner. (Spreading is mostly drawing out crosswise instead of lengthwise.)

3. *Upsetting* is the process of increasing the cross-sectional area of a given portion or possibly of the whole piece. (And, of course, this decreases the length.)

4. *Welding* is the process of joining two surfaces by fusion when they are properly heated and (in hand-forging practice) hammered together.

5. *Forging* consists of all the operations needed to form the required shape and size. It may include one or more of the foregoing operations and also *bending*, *twisting*, *heading*, etc.

Most general-purpose machine shops are equipped with a forge (gas forge usually) for heating the metal and also with

tools used in the forging operations. The forge may be used also for heating steel for hardening and tempering, and near by are two quenching tanks, containing water and oil, respectively. Ash cans make suitable quenching tanks for most small jobs.

297. The Gas Forge.—The proper mixture of gas and low-pressure compressed air under combustion in a gas forge (or gas furnace) gives an intense heat. The *mixture* of gas and air

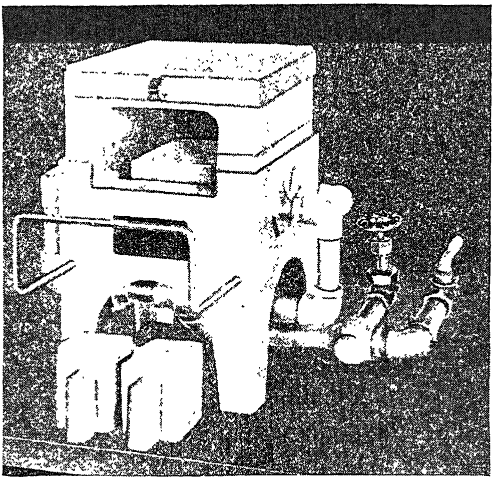


FIG. 244.—Convertible bench forge. Has removable cover brick, and front and back bricks easily handled by tongs. Has adjustable rack for work support. May be used as oven furnace for heat treating carbon or high-speed steel. (*Courtesy of American Gas Furnace Company, Elizabeth, N. J.*)

that will give the best combustion, as well as the amount of this mixture that will give the degree of heat desired, is easily regulated and controlled. A type of gas forge commonly used in toolmaking departments and in general-purpose machine shops is shown in Fig. 244. The air, under pressure of approximately 1 lb. per square inch, comes to the air cock (lever handle). The gas from the main comes to the gas valve (wheel handle). The mixture required, regulated by means of the two handles, goes through the mixture pipes and the four

burners (two on each side) to the combustion chamber. The burners are arranged to give an intense heat with very little air impinging on the work. Too much air oxidizes the surface of the steel and causes a heavy coating of iron oxide (scale). The combustion chamber is of specially molded refractory material (hard-burned firebrick) and will last for years.

To light the forge:

1. Light a suitable piece of paper, and with a pair of tongs place it in the combustion chamber.

2. Turn on the air (lever handle) about halfway—air cock handle about 45 deg.

3. Keep your face away from the opening in the forge (*safety first*), and turn on the gas slowly until the "combustion roar" is heard.

4. Adjust the air cock and gas valve to give the proper flame of the force desired. A soft bluish flame at the burners is considered good combustion. With very little practice the right mixture is regulated for low heat or high heat, as desired.

An improved type of gas forge or furnace, with a special feature called *Single Valve Ratio Control*, as manufactured by the American Gas Furnace Company;[1] or *Single Valve Control Proportional Mixer*, as manufactured by the Chicago Flexible Shaft Company,[2] *automatically* mixes and sends to the burners the correct proportions of gas and air for any degree of heat desired.

The first cost of this furnace is somewhat higher than that of the two-valve-control type illustrated in Fig. 244, but guesswork concerning the mixture is eliminated. It is necessary only to open the gas shutoff valve and then regulate the control valve in the air line to give the degree of heat required. The lighting is similar to that explained above.

298. Tongs and Their Use.—Except in working on the end of a fairly long bar, the piece being forged must be held and

[1] American Gas Furnace Company, Elizabeth, New Jersey.

[2] Stewart Industrial Furnace Division, Chicago Flexible Shaft Company, Chicago, Illinois.

handled with tongs. Many forms and sizes of tongs are used; some are standard; others have jaws especially shaped. The types commonly used are illustrated in Fig. 245.

Flat tongs (*a*, Fig. 245) are used ordinarily for holding flat stock. When they have a small groove running lengthwise in each jaw, they may be used for holding the smaller sizes of rods.

Hollow-bit tongs (*b*, Fig. 245) are used for holding round or square or flat stock. The wide space behind the jaws provides room for the head of a bolt or a similar enlarged part of a

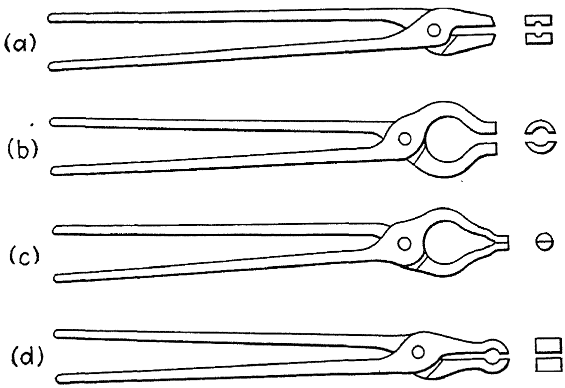


FIG. 245.—Types of tongs commonly used. *a*, Flat; *b*, hollow-bit; *c*, pickup; *d*, link or ring.

given job. It is comparatively easy to fit these tongs (see Art. 299).

Pickup tongs (*c*, Fig. 245) are used more often for handling hot pieces than for holding them during the forging process. When properly shaped, they are not at all clumsy.

Link or *ring tongs* (*d*, Fig. 245) are used primarily for holding curved pieces. Being fairly narrow, the jaws grip closely on a short part of the curve as the piece is being forged. When it is necessary to hold a rod at right angles with the tongs, this type may be used (see Art. 309).

299. Fitting Tongs.—This is a matter of *safety first.* With faulty tongs (*b*, and *c*, Fig. 246) the work is likely to fly out, with consequent serious injury to the smith or someone near by. Use tongs of approximately the right size, and *be sure they are properly fitted.* To fit the tongs, heat the jaws red hot, and hammer them tight on the work, as in *a*, Fig. 246. To keep the handles the right distance apart in fitting the jaws, pinch a suitable piece of steel between them close to the joint. If the handles are too far apart, bend them by hammering close to the joint.

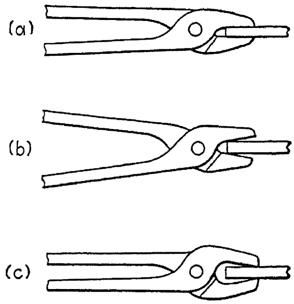


FIG. 246.—Holding work in tongs. *a*, Right; *b*, wrong; *c*, wrong.

300. The Anvil.—The usual anvil weighs around 150 lb. and stands about 26 in. high. To bring it to this height, it is fastened on a cast-iron base or a heavy wooden block. Many prefer the wood on account of its greater resiliency.

The body *A* (Fig. 247) and the horn *B* are of wrought iron or a special grade of steel. The face *C* is a tool steel plate, ½-in. or more thick, welded to the body. The face is heat-treated (hardened and tempered) to resist the sledge-hammer blows on the work being forged. It is smooth and practically flat, being slightly convex crosswise. The horn is primarily for the purpose of bending curved parts. The base *D* of the horn has a flat top on which the work is laid when it is being cut with a chisel. The corners of the face next to the horn are slightly rounded for about 4 in., as at *E*. The round hole *F* (through the tail of the anvil) is the *pritchel hole.* It is useful for punching holes and for heading and also for bending the smaller sizes of rods. The square hole *G* is the *hardie hole* for

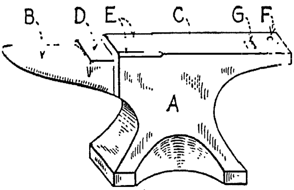


FIG. 247.—Anvil.

receiving various tools with square shanks (see *e*, Fig. 250, also *a* and *b*, Fig. 251). The position of the smith is in front of the anvil with the horn at his left.

301. The Swage Block.—This forge-shop tool (*a*, Fig. 248) is used for many squaring, sizing, heading, bending, and forming operations. It is 1 ft. or more wide and may be used either flat or edgewise in its stand.

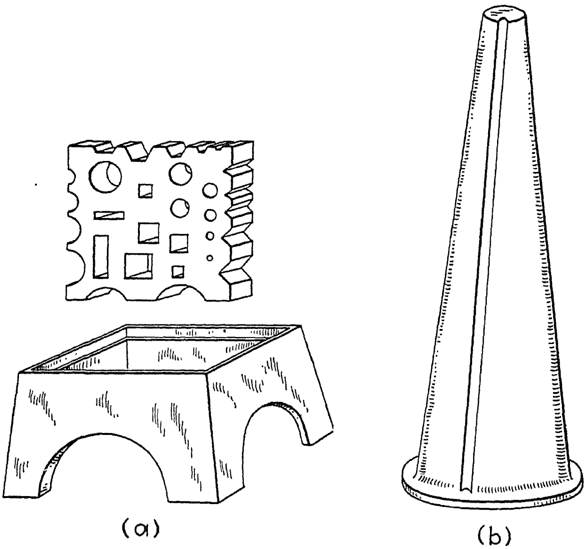


FIG. 248.—*a*, Swage block and stand; *b*, forge-shop cone.

302. The Forge-shop Cone.—The forge-shop mandrel or cone (*b*, Fig. 248) is used for truing and sizing rings. It is made in various sizes, the average being about 3 ft. high and tapering from 2 in. at the top to 12 in. at the base.

303. The Vise.—A vise is a necessity in forge-shop practice. When using a vise and hammer, hammer the work, not the vise! Do not hold a heavy piece of hot metal in the vise, for to do so will destroy the temper of the vise jaws.

304. Hammers.—The face of a hammer is used for ordinary hammering; the peen for drawing out, riveting, scarfing, etc.

The familiar ball-peen hammer (*a*, Fig. 249) is much used by the smith. The weight depends upon his choice for the job at hand and may vary from 1 to 2½ lb. The heavy cross-peen hammer (*b*, Fig. 249) is really a one-hand sledge and weighs from 2½ to 4 lb.

The sledges (*c* and *d*, Fig. 249) are handled by the smith's helper. The lighter type *c* weighs 10 or 12 lb. and usually has a straight peen. In use it is raised head high for a snappy blow. The double-faced sledge *d*, which weighs up to 20 lb., is for heavier striking with a full overarm swing.

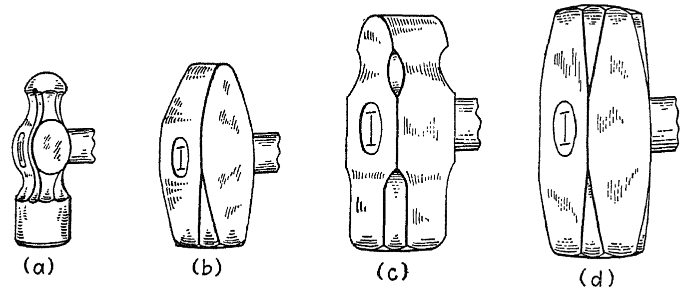


FIG. 249.—Hammers and sledges. *a*, Ball-peen hammer; *b*, heavy hammer, or hand sledge, with cross peen; *c*, sledge with straight peen; *d*, swing sledge, with two faces.

Handles, of course, vary in size and length. On sledges they are about 26 in. long. It is imperative that handles for hammers and sledges are fitted tight and wedged (see page 280). *Never use a hammer or sledge with a loose head or a split handle.*

305. Anvil Tools and Their Use.—For a considerable part of anvil work the piece itself is struck, either by the smith with his hammer or by the helper with his sledge. In the latter case the smith directs the helper's stroke by himself striking a light blow where he wants the sledge to land and a light blow on the anvil as a signal to stop striking. Frequently, however, special anvil tools (Figs. 250 and 251) are held by the smith and struck by the helper. Some anvil tools are made in pairs (Fig. 251),

the upper one fitted with a handle, the lower one provided with a square shank that fits the hardie hole.

The old-time blacksmith used discarded buggy spokes for anvil-tool handles. A smiliar type of handle is still used. It should never be wedged in the eye like a hammer handle but should project an inch or more through the eye, as shown in Figs. 250 and 251. This is for safety; the loosening of the tool on the handle will be noticed at once.

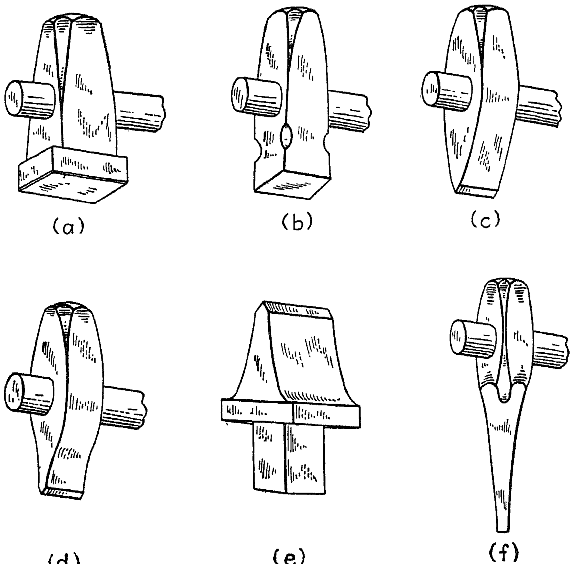


FIG. 250.—Anvil tools. *a*, Flatter; *b*, set hammer or square set; *c*, cold chisel; *d*, hot chisel; *e*, hardie; *f*, punch.

The flatter (*a*, Fig. 250) is used for flattening and smoothing surfaces, removing hammer marks, etc. The face is smooth, about 2½ in. to 3 in. square, with rounded edges.

The set hammer (*b*, Fig. 250) is used for finishing corners in shouldered work where the flatter would be inconvenient. It is also used for drawing out. It is about 1¼ in. square and is made in both square- and rounded-corner types.

The cold chisel (*c*, Fig. 250) is used for nicking cold stock preparatory to breaking off. The bar is nicked all around.

To break off the pieces, the bar is held down on the anvil with the sledge, with the nick on the far edge of the anvil, while the smith strikes a sharp blow on the projecting part. By another method the bar is tightened in the vise with the nick at the edge, and, with the projecting part covered by the end of a pipe sufficiently long to give a good leverage, the pipe is given a sudden snappy jerk.

The hot chisel (*d*, Fig. 250) is thinner than the cold chisel. Hot metal cuts quite easily. Do not use a hot chisel over the face of the anvil; cut either at the edge of the anvil or on a

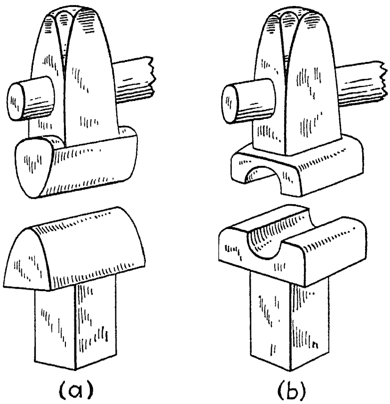


FIG. 251.—Anvil tools. *a*, Top and bottom fullers; *b*, top and bottom swages for round work.

fairly heavy copper strip placed on the base of the horn of the anvil. To cut off square, tip the chisel a trifle, enough to have one face perpendicular.

The hardie (*e*, Fig. 250) also is used for cutting hot metal. Place the work in position on the cutting edge, and strike with hammer or sledge. Use judgment in the force of the blow.

The punch (*f*, Fig. 250) is used for making a hole in hot metal by driving the punch halfway through from each side. First, make adequate center punch marks on opposite sides of the work to indicate clearly, when the metal is red hot, where the hole is to be. Second, drive the punch halfway through from one side. Third, turn the work over, locate the second center

punch mark over the center of the pritchel hole or the hardie hole, and drive the punch to meet the first half of the hole. Fourth, make the hole the required size.

It is often convenient to use the blacksmith's punch to enlarge a given hole or to make a square, oval, or other shape of hole.

Fullers and swages (Fig. 251) are made in many sizes, the top part fitted with a handle and the bottom part held in the hardie hole. Most swages are shaped for finishing cylindrical parts, but other shapes, such as square, hexagonal, etc., are used. Top swages may be used with the swage block (Fig. 249).

Fullers are used for finishing round corners, for making grooves, and frequently for drawing out and for spreading. It is often convenient to use a bottom swage and a top fuller to bend small plates or similar parts.

Questions on Forge-shop Tools

1. Name five forging operations.
2. What is the combustion mixture in a gas forge?
3. What kind of handle has the gas valve?
4. What are the successive steps in lighting the gas forge?
5. Name three kinds of tongs.
6. What is likely to happen when faulty tongs are used?
7. Which hole in the anvil is square?
8. Which side is the front of the anvil?
9. How is the lighter type of sledge used? The heavier type?
10. What is the smith's signal to his helper to stop striking?
11. Are the handles of anvil tools such as flatters, chisels, etc., wedged? Give reason.
12. State the procedure of punching a hole through hot metal.
13. Which type of the smith's chisels is used to cut the metal entirely off?
14. For what purpose may the bottom swage and the top fuller be used together?

FORGING PRACTICE

306. Heating.—Wrought iron and steel are the metals commonly forged. They are soft—plastic—when properly heated and will flow easily under the hammer blows. The forging

heat for steel is a light yellow, considerably above the red-hot color; for wrought iron it may be even hotter, almost the so-called "white hot."

NOTE.—It is impossible to describe exactly the proper colors of heats for forging, welding, or hardening. "Cherry red" does not help much; neither does "white hot." Some experience is necessary to learn the right heats. It may be stated, however, that "when the shadows disappear" describes the correct color for hardening (see Art. 289). Also, "bright yellow" or "light yellow" describes forging color about as nearly as possible, and the steel must be practically scintillating or sparking for the welding operation.

The American Gas Furnace Company gives the following table of colors and equivalent degrees of temperature:

Color	°F.	Color	°F.
Faint red	930	Salmon	1550
Blood red	1075	Dark orange	1634
Dark cherry	1175	Orange	1725
Medium cherry	1275	Lemon	1830
Cherry	1375	Light yellow	1975
Bright cherry	1450	White	2200

Proper forging will improve rather than injure the grain of the steel. Do not hammer the work when it is dull red, for to do so will probably cause cracks to develop. Reheat as often as necessary, but remember that each reheating causes extra scale on the surface of the work. Too high a degree of heat will burn and spoil the steel. *Heat the work slowly, thoroughly, and carefully; hammer it quickly.*

307. Drawing Out.—The smaller sections may be drawn out by laying the heated bar across the face of the anvil and striking with a suitable hammer. Use the face of the hammer or either type of peen, whichever will best serve the purpose. The larger sections are drawn out when the work is laid on the horn of the anvil. This will lessen the tendency to spread (see 2, Art. 296). Further to reduce this tendency, the top fuller may be used with the sledge.

To draw out a piece to a *square* or *rectangular* cross section, keep it fairly square or rectangular as it is worked smaller. Give it frequent quarter and half turns, thus: side *A* (Fig.

252), then a quarter turn to side B, half turn to side C (opposite B), quarter turn to side D, then back to side A, and repeat.

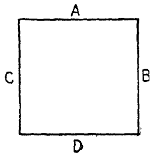


FIG. 252.

To draw out a *round* cross section, from round, square, or other shape, *first draw the piece out square*, as explained above, to about the thickness of the shape (cylindrical or pointed) desired. Then hammer the corners to make it approximately octagonal, and finally round it. If this procedure is not followed, cracks will probably develop.

308. Shoulders.—When a piece is to be drawn out to a shoulder on one side only and the stock is not too large, it may be forged with a hammer as shown in *a* (Fig. 253). The shoulder location point of the heated metal is placed directly over the inside edge of the anvil. If it spreads too much as it is being drawn out, hammer the edges occasionally to keep working toward the general shape required in the finished forging. For the larger bars, as in *b* (Fig. 253), the set hammer

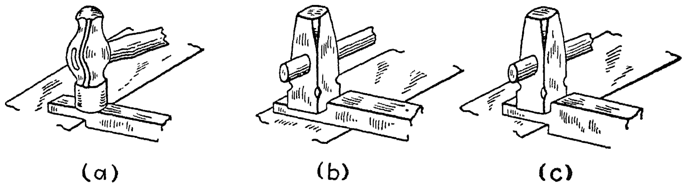


FIG. 253.—Shouldering. *a*, Shoulder on one side, smaller stock; *b*, shoulder on one side, heavier stock; *c*, double shoulder.

and sledge may be used for both drawing out and shouldering. Hold the work well forward on the anvil, thus avoiding a tendency to form a shoulder on the underside.

To forge shoulders on both sides, place the heated bar on the anvil with the shoulder location point directly over the inside edge of the anvil, and hold the set hammer with its edge directly above, as shown in *c* (Fig. 253). Give the work a half turn occasionally to keep both sides even.

To draw out a bar uniformly to a shoulder—for example, a cylindrical or a square reduction along the center line of a given bar—the reduced portion is first forged square (Art. 307)

to about the size required. This is done in a manner similar to that for the shoulder on both sides, (*c*, Fig. 253). That is, it is forged with the shoulder at the edge of the anvil and the edge of the set hammer directly above. If the reduced portion is to be cylindrical, remember that it is first forged square, then octagonal, and finally rounded.

309. Upsetting.—To enlarge the cross-sectional area of a given portion of a piece of steel or wrought iron, heat slowly and thoroughly until almost white hot that part of the bar (usually one end) at which the enlargement is wanted. If the bar is fairly long, the cold part may be grasped in the hands and the heated end struck against the face or side of the anvil. Shorter pieces are held vertically on the anvil with link tongs (*d*, Fig. 245) and the colder end struck with hammer or sledge. In either case heavy blows and perhaps occasional straightening are necessary.

310. Bending.—Be sure that the bend is started in the right place. It is especially true that the sharper bends must be started right, and one or more center-punch marks may be used to show where the inside corner is to be.

The longer bends—rounder corners—are made over the horn; the sharper bends, over the slightly rounded corners of the anvil face (*E*, Fig. 247). In either case, except in rods smaller than 3/8 or 1/2 in., the helper usually presses the piece down hard on the face or horn of the anvil with his sledge while the smith hammers down the projecting end. The smaller rods may be bent while they are held in the vise without overheating or otherwise injuring the vise.

311. Length of Stock for Bending.—Sometimes for an angle bend, and almost always for a curve bend (as for an eye, a ring, or a link), it is necessary to know the length of the stock needed. This is always calculated as the *length of the center line.* For example, in the right-angle bend shown in *a* (Fig. 254), the outside measurement is 11 in., the inside measurement is 10 in. and the length of the center line is 10 1/2 in. (6 1/4 plus 4 1/4 in.). The smith usually allows a little extra length to cut off one end or the other, perhaps both.

The length of stock to cut off for the ring (*b*, Fig. 254) is the length at the center line. The diameter at the center line, the *mean diameter*, is 6 in.; the mean circumference (6 in. multiplied by 3.14) is 18.84 in., or practically 18⅞ in. for the length to cut off.

312. Bending Rings and Links.—To bend a rod in a circular direction, as for a ring or a link, it may be forged on the horn of the anvil or over a suitable cylindrical piece held in the vise. To avoid deep hammer marks, strike the projecting portion—not directly over the anvil.

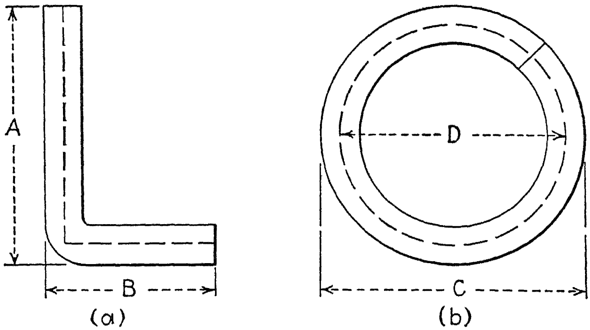


FIG. 254.—*a*, Stock ½ in. Outside measurements: *A* equals 6½ in., *B* equals 4½ in. Length of center line equals 10½ inches. *b*, Stock ½ in. Outside diameter *C* equals 6½ in.; mean diameter *D* equals 6 in.; mean circumference equals 18.84 in.

In forging rings or links, the stock is cut off the proper length. This is the length of the mean circumference (*b*, Fig. 254) plus the little the smith may allow for squaring the ends just before closing. If the piece is to be welded, an amount equal to about one-half the diameter of the stock is allowed to make up for the waste due to scaling.

Rings and links are usually welded (see Art. 314). Cut off the stock, upset the ends, forge the scarves, and bend the piece. Curve the ring by bending approximately halfway from each end, finally overlapping the scarfed ends. Curve the link by bending first in a U shape with ends even; then bend the scarfed ends to overlap. Make the weld, and finish to shape and size.

313. Bending Eyes.—When it is required to forge an eye on the end of a rod or bar, enough of the piece to form the eye is bent at a right angle having nearly a square corner, as shown in *a* (Fig. 255). This amount is equal to the mean circumference of the eye, as explained in Art. 311.

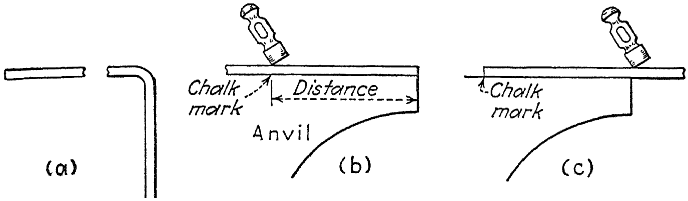


Fig. 255.—Laying off the length for a bend.

When a center-punch mark to lay off the distance would injure the finished job, the smith makes a chalk mark on the anvil a distance from the end of the anvil equal to the length required for the eye. When the work is heated, ready to bend, it is laid along the anvil with the end of the work flush with the end of the anvil, as in *b* (Fig. 255), or flush with the chalk mark, as in *c* (Fig. 255), whichever is preferred. Then

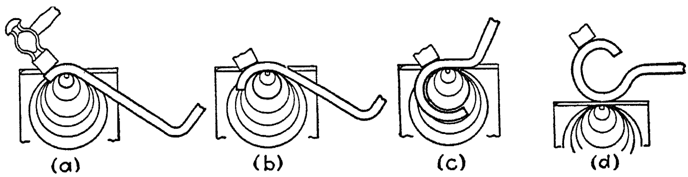


Fig. 256.—Bending an eye. *a*, Starting; *b*, continuing; *c*, nearly ready to close; *d*, closing.

the edge of the hammer is placed on the piece at the point where the bend is to be made, and, with the hammer in this position, both work and hammer are moved to the slightly rounded edge of the anvil; the overhang will be the amount to bend down.

After the right-angle bend is made, the ring is reheated and forged as illustrated in Fig. 256, starting at the end and gradually completing the curve.

314. Welding.—The ends to be joined must first be upset, as in *a* (Fig. 257). This is to provide for the reduction in the cross-sectional area due to hammering the joint in welding. Heat white hot for upsetting, and upset plenty rather than not enough.

After the ends are upset they will have to be *scarfed*, which means forming the ends preparatory to welding. In hand forging, the lap weld[1] is used mostly for round, square, and

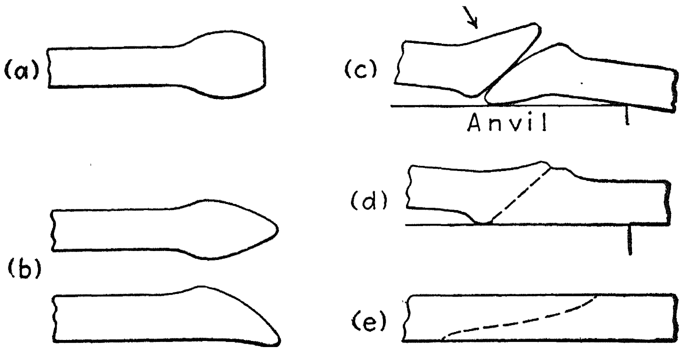


FIG. 257.—Steps in welding. *a*, The upset end; *b*, face and side of scarf; *c*, on anvil ready for first welding blow; *d*, ready for cleaning and finishing; *e*, finished weld.

rectangular bars. In *b* (Fig. 257), the scarfed ends for a lap weld are shown. The upset ends are beveled with the ball peen, and the surfaces to be welded are finished with the face of the hammer. In length, these surfaces are about one and one-half times the thickness of the upset. They are slightly convex, because the first hammer blow should start the weld in the center, and succeeding blows will tend to squeeze out the melted scale and slag. A flux[2] should be used, particularly

[1] Other welds, such as butt weld, V weld, split weld, etc., are not discussed in this book.

[2] A flux (borax or fine, sharp, clean sand) is used in welding iron or steel to lower the melting point of the scale (oxide of iron) so that it will flow off the surface. An oxidized surface will not weld properly. A flux is a necessity in welding steel because the steel would be injured if heated high enough to melt the scale. It also helps to make a good weld in

in welding steel, to promote the fusion of the metal and make a strong weld.

The scarfed ends and the area for a short distance back should be thoroughly heated to a white scintillating heat—both ends the same (see note in Art. 306). As quickly as possible after heating, sprinkle with flux, and hit the bar with a hammer, or tap it on the anvil horn to shake the scale from the welding surfaces. Place the job on the anvil carefully and quickly, with the point of the top scarf overlapping a trifle the heel of the lower scarf, as shown in *c* (Fig. 257). Strike a hard blow in the center, as indicated by the arrow, and work rapidly from the center with hard, snappy blows. Weld the thinner ends as quickly as possible, because they cool very rapidly. Reheat if, in the operation, the joint falls below the welding heat. Finally clean off the scale with a file, and smooth to size *e* (Fig. 257).

Questions on Forging Practice

1. What do you understand by steel *flowing* under the hammer?

2. Which is hotter, cherry or yellow? About how many degrees?

3. Explain how work is drawn to a square cross section.

4. Explain how work is drawn to a round cross section.

5. Explain the method of drawing equal opposite shoulders on the end of a bar.

6. What type of tongs are used to hold a short piece in upsetting?

7. What is meant by the mean circumference of a forged ring?

8. Why does the smith need to know the approximate mean circumference in forging an eye?

9. Explain how the smith uses a chalk mark on the anvil to lay off on the work the length to be bent.

10. After the work is bent at a right angle, how do you proceed to forge the eye?

11. What is the operation of scarfing?

12. What is the purpose of a flux in welding?

13. Why must the work be very hot in order to weld?

14. Why does the smith work so fast in making a weld?

---

wrought iron. Sprinkle the flux on the welding surfaces just before or immediately after removing the work from the furnace.

EXAMPLE OF FORGING, HARDENING, AND TEMPERING

315. Making a Cold Chisel.—Every machinist should know how to forge, harden, and temper a cold chisel or a screw driver. Frequently a special chisel or screw driver is needed or one that is much used needs to be redressed. Directions given here concerning a cold chisel apply as well to a screw driver.

Examine, if possible, a flat chisel, a cape chisel, and a gouge chisel. Make a flat chisel first, it is easier. Refer to Fig. 258.

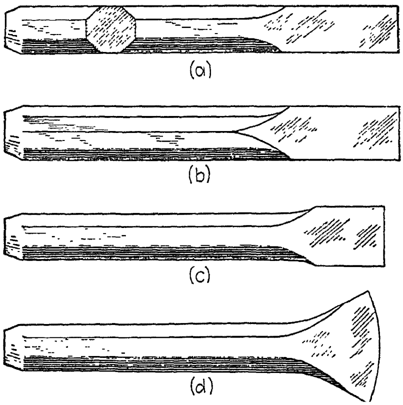


FIG. 258.—Forging a flat cold chisel. *a*, Right. *b*, Wrong, not forged parallel with flat of stock. *c*, Right, edges kept parallel and about the right width. *d*, Wrong, forged fan shape.

Chisel steel is octagon shape. Steel suitable for cold chisels and screw drivers is around 80-point carbon which is high enough to hold an edge and low enough to withstand a forging heat. Get a piece, say ⅝-in. octagon, 6 in. long.

Heat the steel a bright red for forging. Allow a sufficient time for the steel to heat thoroughly, making sure the heat has penetrated to the center. This is called "a soaking heat." Heat it slowly in order that the outside will not become overheated before the soaking heat is given. A bright red is not a white heat, a white heat is too hot. No carbon steel can be heated above a bright red without injuring it.

When forging *make the flats of the chisel parallel with opposite flats of the octagon stock.* This is important for the appearance of the completed chisel. Strike only fairly hard blows. It is better to hammer rather lightly at first, until one has a little practice, than to give too heavy blows and get the chisel out of shape. A blacksmith holds his thumb on top of the hammer handle with an easy grasp but with full control. He strikes solid snappy blows with the face of the hammer—not the edge.

Do not forge the end fish-tail shape as shown in d, Fig. 258. After flattening the end somewhat, turn the chisel a quarter turn and holding the shank horizontally hammer the narrow sides to make them parallel. Alternate the blows, four or five on the wider surface, then four or five on the narrow surface, and so on until the chisel is the shape desired. When forging do not allow the heat to get below a dull red, because hammering the steel after it has cooled too much will set up strains that may crack the steel when the chisel is hardened. Reheat the piece two or three times if necessary—not too hot and not too quickly. As the chisel gets thinner on the forged end, be careful when reheating it not to overheat the thin section.

After the chisel is forged the next operation is *annealing.* Reheat to a cherry red (a little hotter than a dull red) and lay aside to cool. The slower it cools the better. Sometimes it may be left in the furnace to cool with the furnace, or it may be buried in ashes or in powdered charcoal. Ordinarily it is sufficient, however, merely to cool in the air. Annealing serves to relieve the strains set up by forging and to give the steel an even, close grain or texture. (Annealing also serves to soften hardened steel, or hard spots that may be in the steel. Manufacturers of steel anneal most of the carbon steel after rolling because rolling acts to a certain extent as does hammering.) Annealing any piece, after forging and *before hardening,* is a very important step and is frequently overlooked.

The next step is *hardening.* Read hints 1, 2, 3, and 4, page 340. Heat the chisel about 2 in. back from the cutting edge to a "cherry red" (hotter than a dull red but not a bright red—the heat for hardening is not so high as for forging).

Give it "a soaking heat" until "the shadows disappear." Be very careful that the fire is not hot enough to overheat and ruin the thin section of the chisel.

The sequence of the following directions, *a*, *b*, and *c*, for *quenching* the chisel is important. Understand each of the directions and the reason therefor thoroughly so that each step may be made carefully and deliberately without further reference to these instructions.

a. When it is exactly the right heat, remove the chisel from the fire and quickly but deliberately dip it in water about ½ in. and keep it moving up and down for another ¼ or ½ in. This movement is to avoid having too sharp a line between the hardened and unhardened portions. If this is not done the piece is liable to crack at the water line. Keep the ½ in. of chisel *in the water* all of the time until it is finally removed.

b. When the ½ in. of the chisel end is black (hardened) plunge the whole chisel under water and quickly back, nearly out, that is out to the ½-in. portion that must be left in the water. Repeat this two or three times until the red is gone from the sides of the chisel and a dull-red streak only remains along the center.

c. Now quickly remove the chisel from the water and plunge it into the tempering oil. Move it around in the oil and allow to cool. This treatment will give a hardened portion ½ in. or more back from the cutting edge, also the sides are hard and the quenching in oil of the center part while red gives a maximum toughness.

If no oil bath is available, cool the chisel slowly by occasionally dipping it in the water.

Tempering is the next operation. When the chisel is cool remove it from the oil bath and with a piece of emery cloth or broken grinding wheel polish the flat back far enough from the cutting edge to see the temper color run. Draw the temper by holding the chisel over a flame, heating the shank and the heavier section of the forged end and allowing the color to run, that is, draw the temper from the shank towards the cutting edge. The color on the cutting end should be

purple flecked with blue (530 to 540°F.), and this color should extend at least ½ in. back. The chisel is now hardened and properly tempered ½ in. back of the cutting edge and may be sharpened several times before the hardened section is ground away.

Screw drivers should be drawn a trifle more than cold chisels. Draw them to blue (540 to 550°F.).

Tempering heat should penetrate all through the steel and the longer the piece is held at the tempering heat the tougher it will be. Toolmakers have learned that in punch and die work it is wise to leave the die in the tempering oil for several hours. It is much less liable to break when in use and the cutting edge lasts longer.

Reasoning as above, it will be found that if, after the color has been drawn on a screw driver or a chisel, it is polished and drawn again and possibly once again, it will greatly improve the toughness and in no way affect the hardness.

Finally grind the chamfer on the head of the chisel and sharpen the cutting edge by grinding two even parallel facets or bevels (see page 300).

Questions on Making a Cold Chisel

1. What content of carbon has the average cold chisel?

2. In what careful manner is the chisel steel heated for forging?

3. What surfaces of the chisel stock are considered when the "flats" of the chisel are forged?

4. What is the next step after forging a screw driver or a chisel?

5. Which has the temper drawn more, the chisel or the twist drill? Why?

6. Besides relieving forging stresses and giving steel an even texture, what does annealing do?

7. What is the remedy if a chisel edge breaks when in use?

8. What is the remedy if a chisel edge bends or flattens when in use?

9. What is likely to happen if the piece being hardened is not kept in motion in the quenching bath?

10. State how properly to harden and temper a cold chisel.

11. How many degrees has the cutting angle of a cold chisel?

APPENDIX

RULES FOR FINDING THE DIAMETERS AND SPEEDS OF PULLEYS

The speeds (r.p.m.) of driving and driven pulleys are to each other *inversely* as their diameters. That is the *speed* of the driving pulley is to the *speed* of the driven pulley as the *diameter* of the driven pulley is to the *diameter* of the driving pulley,

or
$$S:s = d:D$$

This is usually written $DS = ds$ and is the formula for

The Fundamental Rule for Speeds of Pulleys.—The diameter of the driving pulley multiplied by its speed is equal to the diameter of the driven pulley multiplied by its speed.

Knowing any three of the quantities the fourth can be found by substituting values in the proper one of the following equations:

$$DS = ds$$

Then
$$D = \frac{ds}{S} \tag{1}$$

$$S = \frac{ds}{D} \tag{2}$$

$$d = \frac{DS}{s} \tag{3}$$

$$s = \frac{DS}{d} \tag{4}$$

Example.—A pulley 12 in. in diameter is running 220 r.p.m., and is connected by a belt to a pulley 7 in. in diameter. How many r.p.m. will the smaller pulley make?

Solution.—Use equation (3),

$$d = \frac{DS}{s} = \frac{12 \times 220}{7} = \frac{2640}{7} = 377 + \text{ r.p.m. } Ans.$$

Pulley Train.—The principal driving shafts in a shop are called "main lines" and the smaller shafts that carry the pulleys over the machines are called "countershafts." Often the speed must be reduced between the engine and the main line in which case a "jackshaft" carries the speed-reducing pulleys. When motion of one shaft is transmitted to another and from that to a third and so on to any number of shafts, the pulleys that carry the belts, which transmit the motion, make up what is called a pulley train.

Suppose power is transmitted from a pulley on the motor to a pulley on the line shaft. The motor pulley is the *driving* pulley and the line shaft pulley is the *driven.* The power is further transmitted from another pulley (a driving pulley) on the line to a driven pulley on the countershaft and from another *driving*) pulley on the countershaft to the (*driven*) pulley on the machine.

The problem of calculating the speeds, etc., in a pulley train is the same in principle as for two pulleys. Instead of calculating for each pair of pulleys in the train separately, a combination of the different proportions will give the same result. Hence the following:

Rule for Pulley Speeds.—The continued product of the diameters of the driving pulleys and the speed of the first driver is equal to the continued product of the diameter of the driven pulleys and the speed of the last driven pulley.

Example.—A certain line shaft runs at 250 r.p.m. A 15-in. pulley on this shaft is connected by a belt to a 10-in. pulley on the countershaft. From a 12-in. pulley on the countershaft motion is transmitted to the machine. What diameter must the pulley on the machine be to give a spindle speed of 600 r.p.m.?

Solution.—Let x = diameter of pulley on machine.

$$\underset{\cancel{5}}{\cancel{250}} \times 15 \times \cancel{12} = \underset{2}{\cancel{10}} \times x \times \underset{\cancel{12}}{\cancel{600}}$$

$$15 = 2x \quad \text{or} \quad x = 7\tfrac{1}{2} \text{ in. } \textit{Ans.}$$

RULES FOR FINDING THE NUMBER OF TEETH AND VELOCITY OF GEARS

The velocity (r.p.m.) of the driving gear and the follower gear are to each other *inversely* as the numbers of their teeth. That is, the velocity of the driving gear is to the velocity of the follower gear as the number of teeth in the follower gear is to the number of teeth of the driving gear, or

$$V : v = n : N$$

This is usually written $NV = nv$ and is the formula for

The Fundamental Rule for Velocities of Gears.—The number of teeth in the driving gear multiplied by its velocity is equal to the number of teeth of the driven gear multiplied by its velocity.

Knowing any three of the quantities, the fourth can be found by substituting values in the proper one of the following equations.

$$NV = nv$$

Then

$$N = \frac{nv}{V} \qquad (1)$$

$$V = \frac{nv}{N} \qquad (2)$$

$$n = \frac{NV}{v} \qquad (3)$$

$$v = \frac{NV}{n} \qquad (4)$$

Example.—A gear with 40 teeth meshes with a gear having 96 teeth. If the small gear makes 120 r.p.m., what will be the velocity of the larger gear?

Solution.—Use equation (4),

$$v = \frac{NV}{n} = \frac{\overset{5}{\cancel{40}} \times \overset{10}{\cancel{120}}}{\underset{8}{\cancel{96}}} = 50 \text{ r.p.m. } Ans.$$

A compound gear train is a train of gears composed of two or more pairs, or simple trains, of gears. The problem

of calculating velocities, etc., in compound gearing is the same in principle as for two gears. Instead of calculating for each pair of gears in the train separately a combination of the different proportions will give the same result. Hence the following:

Rule for Compound Gear Velocities.—The continued product of the *numbers* of teeth in the driving gears and the velocity of the first driving gear is equal to the continued product of the *numbers* of teeth in the follower gears and the velocity of the final follower gear.

MACHINE FITS

There are four different kinds of cylindrical fits used in machine work, namely: the running (and sliding) fit, the drive fit, the force fit, and the shrink fit.

In most cases, excepting the shrink fit, the hole should be finished to a standard size, the shaft or stem or other part then fitted to it. Babbitted bearings are an exception.

Sliding and Running Fit.—For a sliding or running fit the diameter of the shaft should be enough smaller to allow for a film of oil for lubrication. This allowance, as it is called, depends on (1) the purpose of the bearing, (2) the diameter of the shaft, (3) the length of the bearing, (4) the kind of metal used for each.

For an average length of bearing an allowance of 0.001 in. per inch of diameter of bearing is sufficient. A longer bearing requires usually a trifle more allowance.

The speed of the shaft is a factor in running fits and the necessity of high speeds and close running fits has developed bearing metals such as babbitt, bronze, and hardened steel which are much used in machine construction. Hardened and ground spindles in hardened and ground bearings require very little allowance, likewise hardened and ground shafts in bronze bearings. Unlike metals work best in running fits with the exception of hardened steel. Like metals may be used for sliding fits. The bar of a sliding fit is usually finished lengthwise by drawfiling to give it a longer life and a better appearance.

Drive Fit.—When two pieces are to retain indefinitely a fixed relative position, they may be so held by driving the one in the other as a key in a shaft, or a straight dowel pin in a hole. The allowance depends on the length and cross section or diameter of the bearing surfaces, and the smoothness of the surface and the form of the surrounding part. The longer bearing surfaces and the larger diameter require less allowance; a carefully ground surface will hold better than a turned or filed surface and therefore does not require so much allowance; and a thin or weak pulley hub for example will not stand the driving allowance for a key that might be used in a heavy solid hub.

Force Fit.—When two pieces are to retain permanently a fixed relation a greater allowance than a drive fit may be used and the one part is forced into the other in a screw press or in a hydraulic press. For example, in forcing the axles into locomotive driving wheels, a pressure of 150 tons is not uncommon.

Shrink Fit.—When two pieces are to remain permanently together but the shape of one or both would make it impracticable or impossible to force one within the other, the enveloping piece is heated and thereby expanded sufficiently to slide over the other and then cooled slowly with water. Care must be taken when a piece is to be shrunk on another against a shoulder to prevent the piece shrinking away from the shoulder, leaving a space between. Cool the part against the shoulder first and then gradually away from the shoulder.

FASTENING A BELT

There are many kinds of patented metal belt fasteners in the market. Some of them are quickly applied and very serviceable but none, except perhaps the wire lacing, or the clipper lacing, is as flexible and smooth running as the old-fashioned rawhide belt lacing, which is still very generally used. It is desirable that every boy in the shop should know how to lace a belt.

The tools used comprise a belt punch, a belt awl, a pair of pliers, a try square, and a sharp knife.

General Directions:

1. Cut the ends of the belt *square.*

2. The "grain" side or hair side of the belt should run against the pulleys (the grain side is the smooth side).

3. The lacing should be crossed on the flesh side (outside) of the belt and *not* on the grain side (side toward pulley).

4. The holes in both ends should be exactly opposite.

5. The belt should not be too tight or it will injure both the belt and the bearings.

6. A belt is made of sections of leather lapped and cemented together. The belt should be put on so that the points of the laps will run against the pulley.

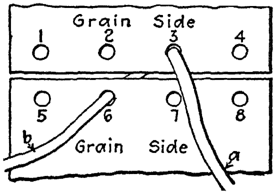


FIG. 259.

The following directions apply particularly to belts from 2 to 5 in. wide.

1. Put the belt around the pulleys and pull tight letting one end lap over the other and note the amount. A good strong pull will indicate the amount the belt must be shortened to give it the proper tension.

2. Lay off the amount to be trimmed with the point of the knife, using the try square as a guide, and then cut straight on the line.

3. Punch holes approximately $\frac{3}{16}$ in. diameter about $\frac{3}{4}$ in. from center to center and not nearer than $\frac{1}{2}$ in. from the end or from the sides.

4. Select a lace, a trifle longer than is required, that will pull fairly easy through the holes. Butt the ends of the belt together with edges flush. Put lace up through holes 3 and 6 from the *flesh* side (see Fig. 259) pulling ends of lace even. Put lace *a* down through 7, up through 4 (and pull tight, using the pliers if necessary), down through 8, up through 4 once more, down through 8 again, up through 3, down through 7, up through 2. Punch a hole $\frac{1}{2}$ in. back of hole 2 and pull the lace through this hole. Cut off lace, leaving tab end of about $\frac{3}{8}$ in. Cut lace nearly half through at the surface of the belt and twist tab end one-half turn, thereby fastening it.

Put lace *b* down through 2, up through 5, down through 1, up through 5, down through 1, up through 6, down through 2, and up through 7. Fasten lace directly back of 7.

Cementing Belts.—Cementing together the lapped ends of a belt, Fig. 260, is considered the best method of fastening. The ends are shaved down as shown so that the lapped portion is not noticeably thicker than the rest of the belt, and the shaved parts should be flat and fit snugly together.

Get two pieces of board about 6 and 12 in. long respectively (for a belt under 4 in. wide) and a little wider than the belt, and arranging the lap about the middle (lengthwise) of the

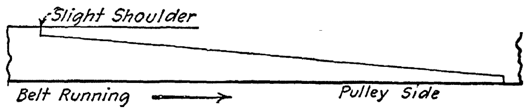


FIG. 260.

longer board lightly clamp at each end. Then fix the lap exactly right, with the edges of the belt flush and straight and set the clamps tight.

Put a piece of paper between the belt and the board and lifting the upper lap apply the cement to both shaved surfaces, rubbing it in well and being careful not to put on too much. Make the joint tight by rubbing out the air pockets, perhaps with the face of a hammer, paying particular attention to the ends and edges. Put a piece of paper over the joint, then the short piece of board, and clamp tightly the two boards with the belt between them. Allow to set for several hours at least, and overnight if convenient. The object of the paper is to keep the boards from sticking to the belt; what little paper cannot be torn off will soon wear off.

GEOMETRICAL PROGRESSION

A series of numbers is said to be in geometrical progression when any number of the series equals the preceding number multiplied by a given constant. For example 2, 4, 8, 16, 32 is such a series and the constant is equal to 2. Any one of these numbers is equal to the preceding number multiplied by 2.

Also, in other words, this constant equals the *ratio* of any number of the series to the next lower number. For example in the series given 16 divided by 8 is equal to 2.

The speed changes and usually the feed changes of most machine tools advance from the slowest to the fastest in geometrical progression. The reason for this is, that in this

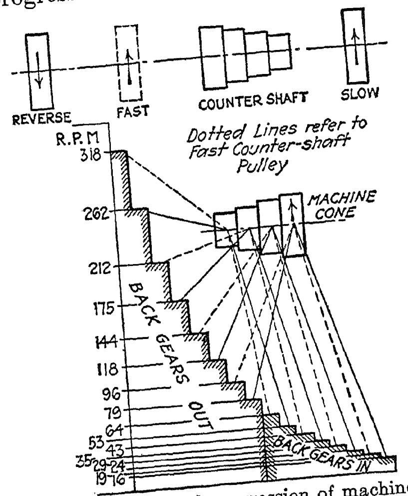


FIG. 261.—Illustrates geometrical progression of machine spindle speeds.

arrangement speeds for the larger diameters of work or cutters (milling cutters, drills, etc.) increase slowly giving a greater number of available speeds, and the speeds for the smaller diameters increase more rapidly which is as it should be as a comparatively small difference in diameter requires a considerable change in speeds. The *ratio* used is usually around 1.2. For example, in Fig. 261, if each speed is multiplied by the ratio 1.21, the quotient will be approximately the next higher speed.

THE VERNIER

The principle of vernier graduations was devised in 1631 by Pierre Vernier. There are two kinds of verniers, direct and retrograde. They are alike in principle but are arranged to

read in opposite directions. The direct vernier is used in various machine-shop measuring instruments and is here described.

A short scale is employed in connection with the regular graduated divisions of the measuring instrument for indicating parts of these divisions. It is so graduated that the space occupied by a certain convenient number of divisions is equal to the space occupied by *one less* than that number of divisions on the true scale of the instrument. To illustrate the vernier principle: Lay off a distance of 9 in. on each of two strips of paper or cardboard and on one divide this distance into 9 equal parts and on the other into 10 equal parts to represent

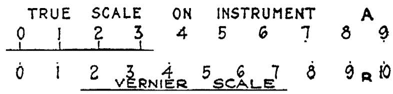


FIG. 262.

respectively the true scale and the vernier scale (see Fig. 262). It will be observed that the *difference in length* between these divisions is equal to one-tenth *of the division on the true scale* or $\frac{1}{10}$ in. (9 in. divided into 9 equal parts; each part equals 1 in. 9 in. divided into 10 equal parts; each part equals 0.9 in. The difference is equal to 1 in. minus 0.9 equals 0.1 in.)

The difference between one division on the true scale and one division on the vernier scale is always equal to a division of the true scale divided by the number of divisions on the vernier scale (in the above case 1 in. divided by 10 equal $\frac{1}{10}$ in.). As a further example: suppose 25 divisions on the vernier scale occupy the same space as 24 divisions on a true scale, the difference between these two divisions is equal to one twenty-fifth ($\frac{1}{25}$) of the true scale division, whatever its length may be.

It will be observed that if the zero mark on the vernier scale (Fig. 262) coincides with the zero mark on the true scale, and the vernier scale is moved so that "1" coincides with a line above, it has moved $\frac{1}{10}$ in. If "2" is exactly opposite a line above, it has moved $\frac{2}{10}$ in. If it is moved so that "5" coincides with a line above it has moved $\frac{5}{10}$ in.; and so on.

The distance that the zero on the vernier has been moved *beyond the preceding line* on the true scale will be indicated by the number of the line *on the vernier scale* that coincides with the line above.

The Vernier Caliper (Fig. 263), Depth Gauge, Height Gauge, and Gear-tooth Vernier

All of these instruments read in exactly the same way. The bar of the instrument is graduated into *fortieths* of an inch. Each inch is numbered and each tenth of an inch is numbered. On the sliding jaw is a vernier scale graduated into 25 parts

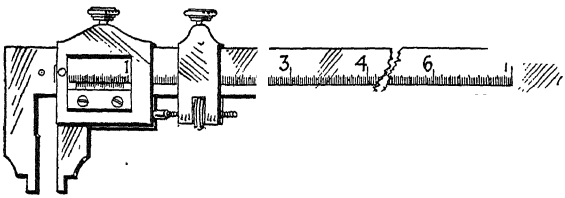


FIG. 263.

numbered, 0, 5, 10, 15, 20, 25. The 25 parts on the vernier correspond, in extreme length, with $^{24}\!/_{40}$ in. on the bar, consequently each division on the vernier is smaller than each division on the bar by one twenty-fifth of $\frac{1}{40}$ in. or one thousandth part of an inch. To read the distance the caliper is open, commence by noticing how many inches, tenths, and fortieths the zero point on the vernier has been moved from the zero point on the bar. [The best way of expressing the value of the division on the bar is to call the tenths, one hundred thousandths (0.100) and the fortieths, twenty-five thousandths (0.025).] Now find on the vernier scale the number of the divisions which is exactly opposite to a line on the bar and this will be the number of thousandths added to the distance read off on the bar.

Referring to Fig. 264 it will be observed, first, that the preliminary reading is one inch, four-tenths and one-fortieth which is equal to one inch and four hundred twenty-five thousandths (1.425), next that the eleventh division on the vernier scale

coincides with the line on the regular scale above, and therefore eleven thousandths more must be added to the one inch and four hundred twenty-five thousandths which will make the reading one inch four hundred thirty-six thousandths total.

In making inside measurements with the 6-in. vernier two hundred and fifty thousandths of an inch, and with the 12-in.

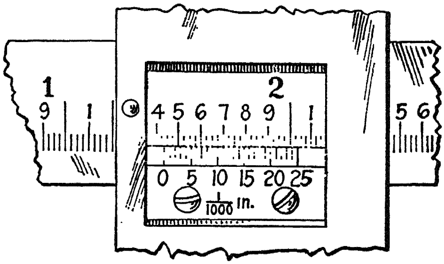


FIG. 264.

and larger verniers, three hundred thousandths of an inch, should be added to the apparent reading of the vernier side for the space occupied by the caliper points.

Vernier for Universal Bevel Protractor (Fig. 265)

The vernier indicates every five minutes (5 min.) or one-twelfth of a degree. Twenty-four spaces on the vernier scale equal in extreme length 23 double degrees. Thus, the difference between the space occupied by two degrees on a regular scale and the space of the vernier scale is equal to one twenty-fourth of two degrees or one-twelfth of one degree (or five minutes).

To Read the Protractor Setting.—Read off directly from the true scale the number of whole degrees between 0 on this scale and the 0 of the vernier scale. Then count, in the same direction, the number of spaces from the zero on the vernier scale to a line that coincides with a line on the regular scale; multiply this number by 5 and the product will be the number of minutes to be added to the whole number of degrees.

For example: In Fig. 265 the 0 on the vernier has moved 12 whole degrees to the right of the 0 on the regular scale

and the 8th line on the vernier coincides with a line upon the regular scale as indicated by *. Multiplying 8 by 5, the product, 40, is the number of minutes to be added to the whole

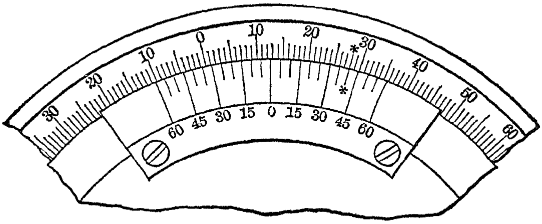


FIG. 265.

number of degrees, thus indicating a setting of 12 degrees and 40 minutes (12° 40′).

MICROMETER WITH VERNIER FOR TEN-THOUSANDTH GRADUATIONS

The reading in ten-thousandths of an inch is obtained by means of vernier graduations on the hub of the micrometer.

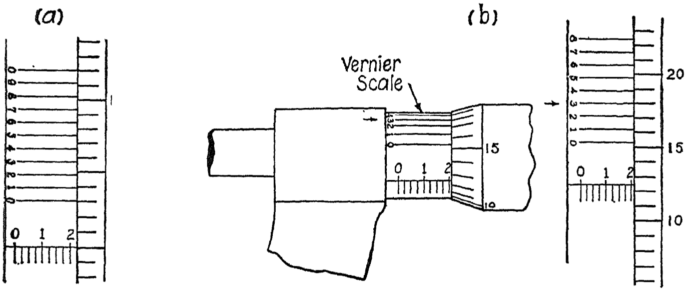


FIG. 266.—Ten-thousandths micrometer. (*a*) Development of vernier scale, and beveled edge of thimble—ten spaces on vernier equals nine spaces on bevel. (*b*) Shows micrometer reading .2123 (.212 + .0003) since the "3" line on the vernier coincides with a line on the bevel.

These divisions are 10 in number and occupy the same space as 9 divisions on the thimble. (For convenience in reading, the graduations are numbered 0, 1, 2, 3, 4, 5, 6, 7, 8, 9, 0.)

Accordingly when a line on the barrel coincides with the first line of the vernier, the next two lines to the right differ

from each other one-tenth of the length of a division on the thimble and indicate one-tenth of a thousandth, the next lines differ by two-tenths, etc.

To Read the Micrometer.—Note the thousandths as usual and the tenths of a thousandth to add will be indicated by the number of the line in the *vernier* scale which coincides with a graduation on the barrel. In *b*, Fig. 266, the reading is 0.2123. It will be noted that without looking at the vernier reading it may be called a scant 0.212½, but with the 3 line in the vernier scale coinciding with a line on the thimble the exact reading is obtained.

SCREW-THREAD MICROMETER, FIG. 267

The distinctive feature in the construction of this micrometer is that the end of the movable spindle is pointed and the fixed end or "anvil" is V shaped. Enough is taken from the end of the point and the bottom of the V is carried down low enough, so that they will not rest on the bottom or top of the thread to be measured but on the cut surface. As the thread itself is measured, it will be seen that the actual outside diameter of the piece does not enter into consideration.

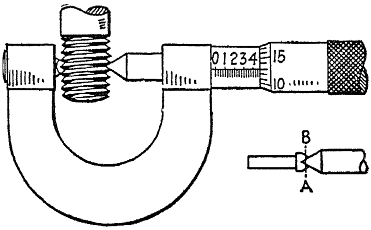


FIG. 267.

As one-half of the depth of the thread from the top is measured on each side, the diameter as indicated by the caliper is the pitch diameter,[1] that is, the full size of the screw less the depth of one thread. When the spindle point and the anvil points are in contact the 0 represents a line drawn through the

[1] *Pitch diameter* of a thread is equal to the nominal outside diameter less the depth of one thread and may be found as follows:

Depth of V thread equals 0.866 ÷ No. of threads per inch.

Depth of Am. Std. thread equals 0.6495 ÷ No. of threads per inch.

Depth of Whitworth thread equals 0.640 ÷ No. of threads per inch.

The pitch diameters for the various sizes of machine screws and standard screws are given in Tables 7, 14, and 15.

plane AB and if the caliper is opened, for example, to 0.463 in. it represents the distance of the two planes, 0.463 in. apart.

The screw-thread micrometer has certain limitations. The given V on the anvil may be used for only a few pitches, for example, one size is used for 8 to 14 threads, the next for 14 or 20 threads, etc. Also the reading is somewhat distorted owing to the slant angle of the thread, and consequently for accuracy the micrometer should be set to a standard thread plug of the given size. This tool is more useful as a gauge than as a measuring tool. The three-wire method has proved more satisfactory as a means of measuring threads.

THREE-WIRE METHOD

The derivation of the constants $3G$ and $1.5155/n$ in the formula for the three-wire method of measuring American Standard screw threads.

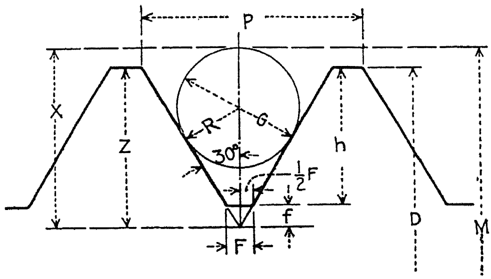


FIG. 268.

p = pitch = $\dfrac{1}{\text{no. of threads per in.}}$ $\dfrac{1}{n}$

h = depth of thread = $0.6495p$.

F = basic flat = $\frac{1}{8}p$. $\frac{1}{2}F = \frac{1}{16}p$

f = basic truncation

Angle of thread = 60 deg.; ½ 60 deg. = 30 deg.

Let $p = 1.000$

By construction: $M = D + 2X - 2Z$ (opposite threads not shown)

$$X = R + \frac{R}{\sin 30^\circ} \qquad (\sin 30^\circ = 0.5)$$

$$2X = 2R + \frac{2R}{0.5} = 6R = 3G \text{ or } (3 \times \text{dia. of wire})$$

$$\begin{aligned} Z = h + f &= (0.6495p) + (\tfrac{1}{16}p \times \cot 30^\circ) \\ &= (0.6495p) + (0.0625p \times 1.7321) \\ &= (0.6495p) + (0.10825p) \\ &= 0.75775p \end{aligned}$$

$$2Z = 1.5155p \text{ or } \left(\text{since } p = \frac{1}{n}\right) = \frac{1.5155}{n}$$

Substituting the values of $2X$ and $2Z$ in making a formula

$$M = D + 3G - \frac{1.5155}{n}$$

LIST OF TABLES

Table 1.—Decimal and Millimeter Equivalents of Fractional Parts of an Inch

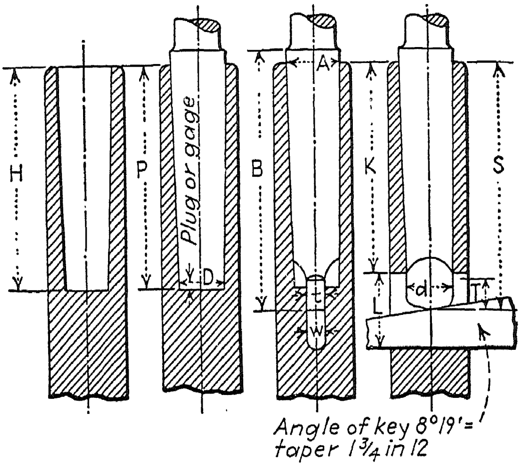
Inches	Inches	Millimeter	Inches	Inches	Millimeter
1/64	0.01563	0.397	33/64	0.51563	13.097
1/32	0.03125	0.794	17/32	0.53125	13.494
3/64	0.04688	1.191	35/64	0.54688	13.890
1/16	0.0625	1.587	9/16	0.5625	14.287
5/64	0.07813	1.984	37/64	0.57813	14.684
3/32	0.09375	2.381	19/32	0.59375	15.081
7/64	0.10938	2.778	39/64	0.60938	15.478
1/8	0.125	3.175	5/8	0.625	15.875
9/64	0.14063	3.572	41/64	0.64063	16.272
5/32	0.15625	3.969	21/32	0.65625	16.669
11/64	0.17188	4.366	43/64	0.67188	17.065
3/16	0.1875	4.762	11/16	0.6875	17.462
13/64	0.20313	5.159	45/64	0.70313	17.859
7/32	0.21875	5.556	23/32	0.71875	18.256
15/64	0.23438	5.953	47/64	0.73438	18.653
1/4	0.25	6.350	3/4	0.75	19.050
17/64	0.26563	6.747	49/64	0.76563	19.447
9/32	0.28125	7.144	25/32	0.78125	19.844
19/64	0.29688	7.541	51/64	0.79688	20.240
5/16	0.3125	7.937	13/16	0.8125	20.637
21/64	0.32813	8.334	53/64	0.82813	21.034
11/32	0.34375	8.731	27/32	0.84375	21.431
23/64	0.35938	9.128	55/64	0.85938	21.828
3/8	0.375	9.525	7/8	0.875	22.225
25/64	0.39063	9.922	57/64	0.89063	22.622
13/32	0.40625	10.319	29/32	0.90625	23.019
27/64	0.42188	10.716	59/64	0.92188	23.415
7/16	0.4375	11.113	15/16	0.9375	23.812
29/64	0.45313	11.509	61/64	0.95313	24.209
15/32	0.46875	11.906	31/32	0.96875	24.606
31/64	0.48438	12.303	63/64	0.98438	25.003
1/2	0.5	12.700	1	1.00000	25.400

TABLE 2.—CUTTING SPEEDS LATHE WORK, DRILLS, MILLING CUTTERS

Formulas: $C.S. = 0.26D \times \text{r.p.m.}$ and $\text{r.p.m.} = \dfrac{C.S.}{0.26D}$

Dia.	Cutting speeds in feet per minute																
	20	30	40	50	60	70	80	90	100	110	120	130	140	150	160	170	180
	Revolutions per minute																
¼	306	458	611	764	916	1070	1222	1376	1528	1681	1833	1986	2139	2292	2462	2615	2780
⅜	204	306	407	509	612	712	814	916	1019	1120	1222	1324	1426	1528	1632	1735	1836
½	153	229	306	382	458	534	612	688	764	840	917	993	1070	1146	1221	1298	1374
⅝	122	183	244	306	366	428	488	550	611	672	733	794	856	917	976	1036	1098
¾	102	153	204	255	306	356	408	458	509	560	611	662	713	764	816	867	918
⅞	87	131	175	218	262	306	350	392	437	480	524	568	611	655	699	742	786
1	76	115	153	191	230	268	306	344	382	420	458	497	535	573	611	649	687
1⅛	68	102	136	170	204	238	272	306	340	373	407	441	475	509	542	576	610
1¼	61	92	122	153	184	214	244	274	306	336	367	397	428	458	489	520	551
1⅜	56	83	111	139	167	194	222	250	278	306	333	361	389	417	444	472	500
1½	51	76	102	127	152	178	204	228	255	280	306	331	357	382	407	433	458
1⅝	47	71	94	118	141	165	188	212	235	259	282	306	329	353	377	400	423
1¾	44	65	87	109	130	152	174	196	218	240	262	284	306	327	349	371	393
1⅞	41	61	82	102	122	143	163	183	204	224	244	265	285	306	326	346	366
2	38	57	76	95	114	134	152	172	191	210	229	248	267	287	306	324	344
2⅛	36	54	72	90	108	126	144	162	180	198	216	234	252	270	288	306	323
2¼	34	51	68	85	102	119	136	153	170	187	204	221	238	255	272	289	306
2⅜	32	48	64	80	97	112	129	145	161	177	193	210	225	241	257	273	290
2½	31	46	61	76	92	106	122	134	153	168	183	199	214	229	244	260	275
2⅝	29	44	58	73	88	102	117	130	146	160	175	189	204	218	233	248	262
2¾	28	42	56	70	83	97	111	125	139	153	167	181	194	208	222	236	250
2⅞	27	40	53	67	80	93	106	119	133	146	159	173	186	199	213	226	239
3	25	38	51	64	76	90	102	114	127	140	153	166	178	191	204	216	226

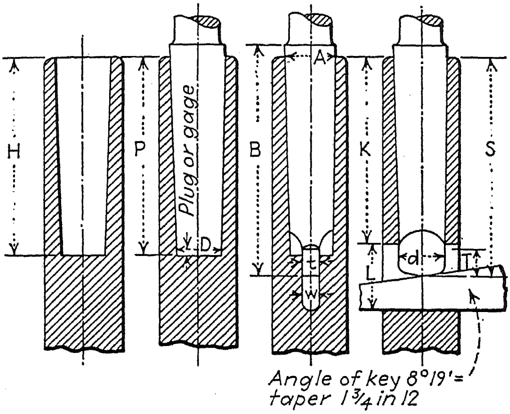
TABLE 3.—MORSE TAPERS



Number of taper	Diameter of plug at small end	Diameter at end of socket	Whole length of shank	Shank depth	Depth of hole	Standard plug depth	Thickness of tongue	Length of tongue	Diameter of tongue	Width of keyway	Length of keyway	End of socket to keyway	Taper per foot
	D	A	B	S	H	P	t	T	d	w	L	K	
0	0.252	0.356	$2\frac{11}{32}$	$2\frac{7}{32}$	$2\frac{1}{32}$	2	$\frac{5}{32}$	$\frac{1}{4}$	0.235	0.160	$\frac{9}{16}$	$1\frac{15}{16}$	0.625
1	0.369	0.475	$2\frac{9}{16}$	$2\frac{7}{16}$	$2\frac{3}{16}$	$2\frac{1}{8}$	$\frac{13}{64}$	$\frac{3}{8}$	0.343	0.213	$\frac{3}{4}$	$2\frac{1}{16}$	0.600
2	0.572	0.700	$3\frac{1}{8}$	$2\frac{15}{16}$	$2\frac{5}{8}$	$2\frac{9}{16}$	$\frac{1}{4}$	$\frac{7}{16}$	$\frac{17}{32}$	0.260	$\frac{7}{8}$	$2\frac{1}{2}$	0.602
3	0.778	0.938	$3\frac{7}{8}$	$3\frac{11}{16}$	$3\frac{1}{4}$	$3\frac{3}{16}$	$\frac{5}{16}$	$\frac{9}{16}$	$\frac{23}{32}$	0.322	$1\frac{3}{16}$	$3\frac{1}{16}$	0.602
4	1.020	1.231	$4\frac{7}{8}$	$4\frac{5}{8}$	$4\frac{1}{8}$	$4\frac{1}{16}$	$\frac{15}{32}$	$\frac{5}{8}$	$\frac{31}{32}$	0.478	$1\frac{1}{4}$	$3\frac{7}{8}$	0.623
5	1.475	1.748	$6\frac{1}{8}$	$5\frac{7}{8}$	$5\frac{1}{4}$	$5\frac{3}{16}$	$\frac{5}{8}$	$\frac{3}{4}$	$1\frac{13}{32}$	0.635	$1\frac{1}{2}$	$4\frac{15}{16}$	0.630
6	2.116	2.494	$8\frac{9}{16}$	$8\frac{1}{4}$	$7\frac{3}{8}$	$7\frac{1}{4}$	$\frac{3}{4}$	$1\frac{1}{8}$	2	0.760	$1\frac{3}{4}$	7	0.626
7	2.750	3.270	$11\frac{1}{4}$	$11\frac{5}{8}$	$10\frac{1}{8}$	10	$1\frac{1}{8}$	$1\frac{3}{8}$	$2\frac{5}{8}$	1.135	$2\frac{5}{8}$	$9\frac{1}{2}$	0.625

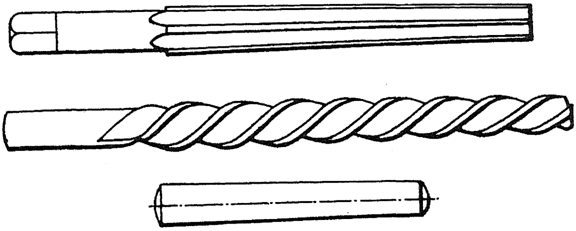
TABLE 4.—BROWN & SHARPE TAPERS

Taper 0.500 in. per foot except No. 10 which is 0.5161 in. per foot



Number of taper	Diameter of plug at small end	Plug depth	Diameter at end of socket	Whole length of shank	Shank depth	Depth of hole	Thickness of tongue	Diameter of tongue	Length of tongue	Width of keyway	Length of keyway	End of socket to keyway
	D	*P*	*A*	*B*	*S*	*H*	*t*	*d*	*T*	*w*	*L*	*K*
4	0.350	1 11/16	0.420	2 3/16	2 3/32	1 13/16	7/32	0.320	11/32	0.228	11/16	1 41/64
5	0.450	2 1/8	0.539	2 21/32	2 9/16	2 1/4	1/4	0.420	3/8	0.260	3/4	2 1/16
6	0.500	2 3/8	0.599	2 31/32	2 7/8	2 1/2	9/32	0.460	7/16	0.291	7/8	2 19/64
7	0.600	2 7/8	0.720	3 1/2	3 13/32	3	5/16	0.560	15/32	0.322	15/16	2 25/32
8	0.750	3 9/16	0.898	4 1/4	4 1/8	3 11/16	11/32	0.710	1/2	0.353	1	3 29/64
9	0.900	4 1/4	1.077	5	4 7/8	4 3/8	3/8	0.860	9/16	0.385	1 1/8	4 1/8
10	1.0446	5	1.260	5 27/32	5 23/32	5 1/8	7/16	1.010	21/32	0.447	1 5/16	4 27/32
11	1.250	5 15/16	1.498	6 25/32	6 21/32	6 1/16	7/16	1.210	21/32	0.447	1 5/16	5 25/32
12	1.500	7 1/8	1.797	8 1/16	7 15/16	7 1/4	1/2	1.460	3/4	0.510	1 1/2	6 15/16

TABLE 5.—TAPER PINS AND REAMERS
(Pratt & Whitney Co.)
Taper = 1/4 in. per foot or 0.0208 in. per inch



Size (No.)	Diameter of small end of reamer	Diameter of large end of reamer	Length of flute	Total length of reamer	Size drill for reamer	Longest limit length of pin	Diameter of large end of pin	Approx. fractional size at large end of pin
0	0.135	0.162	1 5/16	2	28	1	0.156	5/32
1	0.146	0.179	1 9/16	2 3/8	25	1 1/4	0.172	11/64
2	0.162	0.200	1 13/16	2 11/16	19	1 1/2	0.193	3/16
3	0.183	0.226	2 1/16	3	12	1 3/4	0.219	7/32
4	0.208	0.257	2 3/8	3 7/16	3	2	0.250	1/4
5	0.240	0.300	2 7/8	4 1/8	1/4	2 1/4	0.289	19/64
6	0.279	0.354	3 5/8	5	9/32	3 1/4	0.341	11/32
7	0.331	0.423	4 7/16	6 1/16	11/32	3 3/4	0.409	13/32
8	0.398	0.507	5 1/4	7 1/16	13/32	4 1/2	0.492	1/2
9	0.482	0.609	6 1/8	8 1/8	31/64	5 1/4	0.591	19/32
10	0.581	0.727	7	9 1/2	19/32	6	0.706	23/32
11	0.706	0.878	8 1/4	11 1/4	23/32	7 1/4	0.857	55/64
12	0.842	1.050	10	13 3/8	55/64	8 3/4	1.013	1 1/64
13	1.009	1.259	12	16	1 1/64	10 3/4	1.233	1 15/64

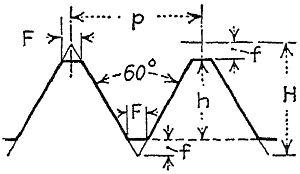
These reamer sizes are so proportioned that each overlaps the size smaller about 1/2 in.

The helical reamers are designed especially for machine reaming on a production basis. They are very free cutting, the chips do not pack in the flutes, and there is a minimum of breakage.

TABLE 6.—TAPERS PER FOOT AND CORRESPONDING ANGLES

Taper per foot	Included angle	Angle with center line	Taper per foot	Included angle	Angle with center line
1/16	0°–18′	0°–09′	1 1/8	5°–22′	2°–41′
1/8	0°–36′	0°–18′	1 1/4	5°–57 1/2′	2°–58 3/4′
3/16	0°–53 1/2′	0°–26 3/4′	1 3/8	6°–33 1/2′	3°–16 3/4′
1/4	1°–11 1/2′	0°–35 3/4′	1 1/2	7°–09′	3°–34 1/2′
5/16	1°–29 1/2′	0°–44 3/4′	1 5/8	7°–45′	3°–52 1/2′
3/8	1°–47 1/2′	0°–53 3/4′	1 3/4	8°–20 1/2′	4°–10 1/4′
7/16	2°–05 1/2′	1°–02 3/4′	1 7/8	8°–56′	4°–28′
1/2	2°–23′	1°–11 1/2′	2	9°–31 1/2′	4°–45 3/4′
9/16	2°–41′	1°–20 1/2′	2 1/4	10°–43 1/2′	5°–21 3/4′
5/8	2°–59′	1°–29 1/2′	2 1/2	11°–53 1/2′	5°–56 3/4′
11/16	3°–10 1/2′	1°–38 1/4′	2 3/4	13°–09 1/2′	6°–34 3/4′
3/4	3°–35′	1°–47 1/2′	3	14°–15′	7°– 7 1/2′
13/16	3°–53′	1°–56 1/2′	3 1/2	16°–35 1/2′	8°–17 3/4′
7/8	4°–10 1/2′	2°–05 1/4′	4	18°–55 1/2′	9°–27 3/4′
15/16	4°–28 1/2′	2°–14 1/4′	4 1/2	21°–37′	10°–48 1/2′
1	4°–46 1/2′	2°–23 1/4′	5	23°–32′	11°–46′

TABLE 7.—AMERICAN STANDARD SCREW THREADS
NATIONAL COARSE (NC) AND NATIONAL FINE (NF)
Thread Dimensions and Tap-drill Sizes



n = no. of threads per inch

p (pitch) $= \dfrac{1}{\text{no. of threads per inch}} = \dfrac{1}{n}$

h (depth) $= 0.649519p = \dfrac{0.649519}{n}$

H (depth, sharp V thread) $= 0.866025p$

$f = \dfrac{H}{8}$ = depth of basic truncation

$F = 0.125p = \dfrac{p}{8}$ = width of basic flat at top, crest or root

National Coarse is the former U. S. Standard for sizes ¼ in. and larger, while for sizes under ¼ in. it is the coarse threads of the former A. S. M. E. machine-screw sizes.

National Fine is the former S. A. E. Standard for sizes ¼ in. and larger, while for sizes under ¼ in. it is the fine threads of the former A. S. M. E. machine-screw sizes.

* American National Standard wood screws are made in same numbers and corresponding body diameters as starred sizes.

Nominal size and No. threads per inch	Major diameter	Pitch diameter	Minor diameter	Commercial tap drill to produce approx. 75 % full thread	Decimal equivalent of tap drill
*0-80	0.0600	0.0519	0.0438	3-64	0.0469
*1-64	0.0730	0.0629	0.0527	53	0.0595
72	0.0730	0.0640	0.0550	53	0.0595
*2-56	0.0860	0.0744	0.0628	50	0.0700
64	0.0860	0.0759	0.0657	50	0.0700
*3-48	0.0990	0.0855	0.0719	47	0.0785
56	0.0990	0.0874	0.0758	45	0.0820
*4-40	0.1120	0.0958	0.0795	43	0.0890
48	0.1120	0.0985	0.0849	42	0.0935
*5-40	0.1250	0.1088	0.0925	38	0.1015
44	0.1250	0.1102	0.0955	37	0.1040
*6-32	0.1380	0.1177	0.0974	36	0.1065
40	0.1380	0.1218	0.1055	33	0.1130
*8-32	0.1640	0.1437	0.1234	29	0.1360
36	0.1640	0.1460	0.1279	29	0.1360
*10-24	0.1900	0.1629	0.1359	25	0.1495
32	0.1900	0.1697	0.1494	21	0.1590
*12-24	0.2160	0.1889	0.1619	16	0.1770
28	0.2160	0.1928	0.1696	14	0.1820
¼-20	0.2500	0.2175	0.1850	7	0.2010
28	0.2500	0.2268	0.2036	3	0.2130
5/16-18	0.3125	0.2764	0.2403	F	0.2570
24	0.3125	0.2854	0.2584	I	0.2720
⅜-16	0.3750	0.3344	0.2938	5-16	0.3125
24	0.3750	0.3479	0.3209	Q	0.3320

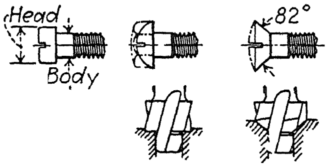
TABLE 7.—AMERICAN STANDARD SCREW THREADS.—(*Continued*)

Nominal size and No. threads per inch	Major diameter	Pitch diameter	Minor diameter	Commercial tap drill to produce approx. 75 % full thread	Decimal equivalent of tap drill
7/16-14	0.4375	0.3911	0.3447	U	0.3680
20	0.4375	0.4050	0.3726	25-64	0.3906
½-13	0.5000	0.4501	0.4001	27-64	0.4219
20	0.5000	0.4675	0.4351	29-64	0.4531
9/16-12	0.5625	0.5084	0.4542	31-64	0.4844
18	0.5625	0.5264	0.4903	33-64	0.5156
⅝-11	0.6250	0.5660	0.5069	17-32	0.5312
18	0.6250	0.5889	0.5528	37-64	0.5781
¾-10	0.7500	0.6850	0.6201	21-32	0.6562
16	0.7500	0.7094	0.6688	11-16	0.6875
⅞- 9	0.8750	0.8029	0.7307	49-64	0.7656
14	0.8750	0.8286	0.7822	13-16	0.8125
1- 8	1.0000	0.9188	0.8376	7-8	0.8750
14	1.0000	0.9536	0.9072	15-16	0.9375
1⅛- 7	1.1250	1.0322	0.9394	63-64	0.9844
12	1.1250	1.0709	1.0168	1 3-64	1.0469
1¼- 7	1.2500	1.1572	1.0644	1 7-64	1.1094
12	1.2500	1.1959	1.1418	1 11-64	1.1719
1⅜- 6	1.3750	1.2667	1.1585	1 7-32	1.2187
12	1.3750	1.3209	1.2668	1 19-64	1.2969
1½- 6	1.5000	1.3917	1.2835	1 11-32	1.3437
12	1.5000	1.4459	1.3918	1 27-64	1.4219
1¾- 5	1.7500	1.6201	1.4902	1 9-16	1.5625
2 - 4½	2.0000	1.8557	1.7113	1 25-32	1.7812
2¼- 4½	2.2500	2.1057	1.9613	2 1-32	2.0312
2½- 4	2.5000	2.3376	2.1752	2 1-4	2.2500
2¾- 4	2.7500	2.5876	2.4252	2 1-2	2.5000
3- 4	3.0000	2.8376	2.6752	2 3-4	2.7500
3¼- 4	3.2500	3.0876	2.9252	3	3.0000
3½- 4	3.5000	3.3376	3.1752	3 1-4	3.2500
3¾- 4	3.7500	3.5876	3.4252	3 1-2	3.5000
4- 4	4.0000	3.8376	3.6752	3 3-4	3.7500

REASONS FOR FINER PITCHES (NATIONAL FINE)

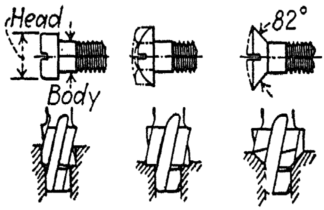
Threads in automobile work are cut in hard tough materials and do not require to be so coarse as threads cut in cast iron. A screw or bolt of a given size and of finer pitch has greater minor diameter and consequently greater strength than a coarse-pitch screw of same size. A fine-pitch screw or nut may be set up tighter and does not shake loose so readily as one of coarse pitch.

TABLE 8.—COUNTERBORE SIZES FOR MACHINE SCREWS



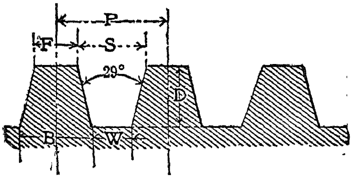
Screw sizes	No. thds. per in. NC	No. thds. per in. NF	Fillister head mach. screw head & body: Bore for head	Fillister head mach. screw head & body: Pilot for body	Round or hex. head mach. screw head & body: Bore for head	Round or hex. head mach. screw head & body: Pilot for body	Flat or oval head mach. screw head & body: Bore for head	Flat or oval head mach. screw head & body: Pilot for body	Tap drills NC	Tap drills NF	Decimal equivalents of tap-drill sizes
0		80	0.096	0.060	0.125	0.060	0.125	0.060		3/64	0.0469
1	64		0.118	0.073	0.156	0.073	0.156	0.073	53		0.0595
		72								53	0.0595
2	56		0.140	0.086	0.187	0.086	0.187	0.086	50		0.0700
		64								50	0.0700
3	48		0.161	0.099	0.203	0.099	0.218	0.099	47		0.0785
		56								45	0.0820
4	40		0.183	0.112	0.234	0.112	0.250	0.112	43		0.0890
		48								42	0.0935
5	40		0.205	0.125	0.250	0.125	0.281	0.125	38		0.1015
		44								37	0.1040
6	32		0.227	0.138	0.281	0.138	0.312	.138	36		0.1065
		40								33	0.1130
8	32		0.270	0.164	0.328	0.164	0.375	0.164	29		0.1360
		36								29	0.1360
10	24		0.313	0.190	0.375	0.190	0.437	0.190	25		0.1495
		32								21	0.1590
12	24		0.357	0.216	0.437	0.216	0.500	0.216	16		0.1770
		28								14	0.1820
1/4	20		0.414	0.250	0.500	0.250	0.562	0.250	7		0.2010
		28								3	0.2130
5/16	18		0.519	0.312	0.625	0.312	0.656	0.312	F		0.2570
		24								I	0.2720
3/8	16		0.622	0.375	0.750	0.375	0.781	0.375	5/16		0.3125
		24								Q	0.3320
7/16	14		0.719	0.437	0.875	0.437	0.937	0.437	U		0.3680
		20								25/64	0.3906
1/2	13		0.820	0.500	1.000	0.500	1.062	0.500	27/64		0.4219
		20								29/64	0.4531

TABLE 9.—COUNTERBORE SIZES FOR CAP SCREWS



Major diameter	No. of threads per inch	Fillister head cap screw head & body		Button or hex. head cap screw head & body		Flat or oval head cap screw head & body		Tap drill	Decimal equivalent of tap drill
		Bore for head	Pilot for body	Bore for head	Pilot for body	Bore for head	Pilot for body		
1/4	20	0.375	0.250	0.500	0.250	0.562	0.250	7	0.2010
5/16	18	0.437	0.312	0.625	0.312	0.656	0.312	F	0.2570
3/8	16	0.562	0.375	0.687	0.375	0.781	0.375	5/16	0.3125
7/16	14	0.625	0.437	0.813	0.437	0.875	0.437	U	0.3680
1/2	13	0.750	0.500	0.875	0.500	0.968	0.500	27/64	0.4218
9/16	12	0.812	0.562	1.000	0.562	1.062	0.562	31/64	0.4844
5/8	11	0.875	0.625	1.062	0.625	1.187	0.625	17/32	0.5312
3/4	10	1.000	0.750	1.312	0.750	1.437	0.750	21/32	0.6562
7/8	9	1.125	0.875	1.375	0.875	.....	.....	49/64	0.7656
1	8	1.312	1.000	1.500	1.000	.....	.....	7/8	0.8750

TABLE 10.—ACME 29-DEG. SCREW THREADS



N = no. of threads per inch

$P = \frac{1}{N}$ = linear pitch $W = .3707P - .0052$

$D = .5P + .01$ $S = .6293P$

$F = .3707P$ $B = .6293P + .0052$

The Acme standard thread is an adaptation of the most commonly used style of worm thread and is intended to take the place of the square thread.

It is a little shallower than the worm thread, but the same depth as the square thread and much stronger than the latter.

The various parts of the Acme standard thread are obtained as follows:

Width of point of tool for screw thread =

$$\frac{0.3707}{\text{no. of threads per inch}} - 0.0052.$$

$$\text{Width of screw or nut thread} = \frac{0.3707}{\text{no. of threads per inch}}$$

Minor diameter =

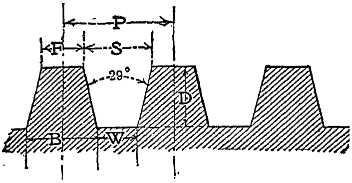
$$\text{Major diameter} - \left(\frac{}{\text{no. of threads per inch}} + 0.020\right)$$

$$\text{Depth of thread} = \frac{1}{2 \times \text{no. of threads per inch}} + 0.010.$$

TABLE OF ACME 29-DEG. SCREW-THREAD PARTS

N	P	D	F	W	S	B
Number of Threads per Inch	Pitch of Single Thread	Depth of Thread	Width of Top of Thread	Width of Space at Bottom of Thread	Width of Space at Top of Thread	Thickness at Root of Thread
1	1.0	.5100	.3707	.3655	.6293	.6345
1⅓	.750	.3850	.2780	.2728	.4720	.4772
2	.500	.2600	.1853	.1801	.3147	.3199
3	.3333	.1767	.1235	.1183	.2098	.2150
4	.250	.1350	.0927	.0875	.1573	.1625
5	.200	.1100	.0741	.0689	.1259	.1311
6	.1666	.0933	.0618	.0566	.1049	.1101
7	.1428	.0814	.0529	.0478	.0899	.0951
8	.125	.0725	.0463	.0411	.0787	.0839
9	.1111	.0655	.0413	.0361	.0699	.0751
10	.10	.0600	.0371	.0319	.0629	.0681

TABLE 11.—ACME 29-DEG. TAP THREADS



N = no. of threads per inch

$P = \frac{1}{N}$ = linear pitch $\quad W = .3707P - .0052$

$D = .5P + .02 \quad S = .6293P + .0052$

$F = .3707P - .0052 \quad B = .6293P + .0052$

The Acme standard tap thread is cut with the same width of tool as the screw thread and the diameter at the root is the same for tap and screw. Clearance at bottom of thread between screw and nut is obtained by boring the nut blank 0.020 oversize.

The outside diameter of the tap is made 0.020 larger than the screw to give clearance between top of screw thread and bottom of nut.

Width of point of tool for tap thread =

$$\frac{0.3707}{\text{no. of threads per inch}} - 0.0052.$$

$$\text{Width of thread} \quad \frac{0.3707}{\text{no. of threads per inch}} - 0.0052$$

$$\text{Diameter of tap} = \text{diameter of screw} + 0.020.$$

Minor diameter =

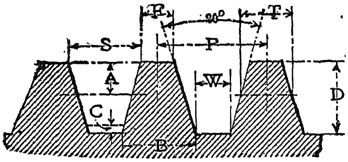
$$\text{diameter of tap} - \left(\frac{1}{\text{no. of threads per inch}} + 0.040.\right)$$

$$\text{Depth of thread} = \frac{}{2 \times \text{no. of threads per inch}} + 0.020.$$

TABLE OF ACME STANDARD 29-DEG. TAP-THREAD PARTS

N	P	D	F	W	S	B
Number of Threads per Inch	Pitch of Single Thread	Depth of Thread	Width of Top of Thread	Width of Space at Bottom of Thread	Width of Space at Top of Thread	Thickness at Root of Thread
1	1.0	.5200	.3655	.3655	.6345	.6345
1$\frac{1}{3}$	.750	.3950	.2728	.2728	.4772	.4772
2	.500	.2700	.1801	.1801	.3199	.3199
3	.3333	.1867	.1183	.1183	.2150	.2150
4	.250	.1450	.0875	.0875	.1625	.1625
5	.200	.1200	.0689	.0689	.1311	.1311
6	.1666	.1033	.0566	.0566	.1101	.1101
7	.1428	.0914	.0478	.0478	.0951	.0951
8	.125	.0825	.0411	.0411	.0839	.0839
9	.1111	.0755	.0361	.0361	.0751	.0751
10	.10	.0700	.0319	0319	.0681	.0681

Table 12.—Brown & Sharpe 29-deg. Worm Threads



N = no. of threads per inch

$P = \frac{1}{N}$ = linear pitch

$D = .6866P$ $\quad A = .3183P$

$F = .335P$ $\quad C = \frac{T}{10}$.

$W = .31P$ $\quad S = .665P$

$T = .5P$ $\quad B = .69P$

Pitch $\dfrac{1}{\text{no. of threads per inch}}$.

Depth of thread $\dfrac{0.6866}{\text{no. of threads per inch}}$.

Width of top of thread $\dfrac{0.335}{\text{no. of threads per inch}}$.

Width of space at bottom $\dfrac{0.310}{\text{no. of threads per inch}}$.

Clearance at bottom of thread $\dfrac{\text{thickness at pitch line}}{10}$.

Width of space at top of thread $\dfrac{0.665}{\text{no. of threads per inch}}$.

Thickness at root of thread $\dfrac{0.69}{\text{no. of threads per inch}}$.

	P	D	F	W	T	A	C	S	B
Number of Threads Per Inch	Pitch of Single Thread	Depth of Thread	Width of Top of Thread	Width of Space at Bottom	Thickness of Thread at Pitch Line	Thread Above Pitch Line	Clearance at Bottom of Thread	Width of Space at Top	Thickness at Root of Thread
1	1.0	.6866	.3350	.3100	.5000	.3183	.05	.665	.69
1¼	.8	.5492	.2680	.2480	.4000	.2546	.04	.532	.552
1½	.6666	.4577	.2233	.2066	.3333	.2122	.0333	.4433	.4599
2	.5	.3433	.1675	.1550	.2500	.1592	.0250	.3325	.345
2½	.4	.2746	.1340	.1240	.2000	.1273	.0200	.2660	.276
3	.3333	.2289	.1117	.1033	.1666	.1061	.0166	.2216	.2299
3½	.2857	.1962	.0957	.0886	.1429	.0909	.0143	.1901	.2011
4	.250	.1716	.0838	.0775	.1250	.0796	.0125	.1637	.1725
4½	.2222	.1526	.0744	.0689	.1111	.0707	.0111	.1478	.1533
5	.2	.1373	.0670	.0620	.1000	.0637	.0100	.1330	.138
6	.1666	.1144	.0558	.0517	.0833	.0531	.0083	.1108	.115
7	.1428	.0981	.0479	.0443	.0714	.0455	.0071	.095	.0985
8	.125	.0858	.0419	.0388	.0625	.0398	.0062	.0818	.0862
9	.1111	.0763	.0372	.0344	.0555	.0354	.0055	.0739	.0766
10	.10	.0687	.0335	.0310	.0500	.0318	.005	.0665	.069
12	.0833	.0572	.0279	.0258	.0416	.0263	.0042	.0551	.0575
16	.0625	.0429	.0209	.0194	.0312	.0199	.0031	.0409	.0431
20	.050	.0343	.0167	.0155	.0250	.0159	.0025	.0332	.0345

TABLE 13.—AMERICAN (N. P. T.) STANDARD TAPER PIPE TAPS
Drill sizes for tapping without reaming

Size of pipe	Threads per inch	Actual inside diameter	Actual outside diameter	Minor diameter small end of tap	Minor diameter small end of pipe and gauge	Tap drill	
						Size	Decimal equivalent
1/8	27	0.270	0.405	0.3145	0.3339	R	0.339
1/4	18	0.364	0.540	0.4043	0.4329	7/16	0.437
3/8	18	0.494	0.675	0.5393	0.5676	37/64	0.578
1/2	14	0.623	0.840	0.6651	0.7013	23/32	0.719
3/4	14	0.824	1.050	0.8751	0.9105	59/64	0.921
1	11 1/2	1.048	1.315	1.1017	1.1441	1 5/32	1.156
1 1/4	11 1/2	1.380	1.660	1.4447	1.4876	1 1/2	1.500
1 1/2	11 1/2	1.610	1.900	1.6828	1.7265	1 47/64	1.734
2	11 1/2	2.067	2.375	2.1578	2.1995	2 7/32	2.218
2 1/2	8	2.468	2.875	2.5617	2.6195	2 5/8	2.625
3	8	3.067	3.500	3.1828	3.2406	3 1/4	3.250
3 1/2	8	3.548	4.000	3.6789	3.7375	3 3/4	3.750
4	8	4.026	4.500	4.1750	4.2344	4 1/4	4.250

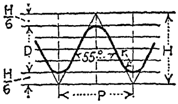
NOTE: Standard pipes and pipe fittings go to 12 in. diameter.

It is frequently necessary to drill holes which are to be tapped for pipes and fittings. The American Standard is used throughout the United States. The "nominal" inside diameter of the pipe is used to designate the size of the pipe. The fact that there is considerable difference in the "nominal" diameter and actual diameter, especially in the smaller sizes, is often confusing.

For five or six threads the outside diameter of the pipe tapers and the diameter at the root of the thread follow the same taper. In this part the threads have full depth. For the next two threads the taper at the root continues, but the outside not being tapered these threads have imperfect tops. The remaining threads are increasingly imperfect on the top and also on the bottom because of the chamfer or bell mouth of the threading die.

In extra strong or double extra strong or hydraulic pipe the additional thickness is on the inside of the pipe and does not affect the thread dimensions. When cutting threads on pipe or pipe fittings set the tool at right angles to the axis of the piece.

TABLE 14.—BRITISH STANDARD WHITWORTH THREADS



P (pitch) $= \dfrac{1}{\text{no. of threads per inch}}$

D (depth) $= 0.6403P = 0.6403$

$H = 0.9605P$

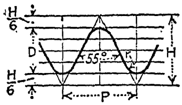
$\dfrac{H}{6} = 0.1600P$

$r = 0.1373P$

n = no. of threads per inch

Nominal diameter of screw	No. of threads per inch	Std. single depth of thread	Effective diameter (pitch diameter)	Commercial tap drill to produce 75% full thread	Nominal diameter of screw	No. of threads per inch	Std. single depth of thread	Effective diameter (pitch diameter)	Commercial tap drill to produce 75% full thread
1/4	20	0.0320	0.2180	13/64	1	8	0.0800	0.9200	7/8
5/16	18	0.0356	0.2769	F	1 1/8	7	0.0915	1.0335	1
3/8	16	0.0400	0.3350	O	1 1/4	7	0.0915	1.1585	1 7/64
7/16	14	0.0457	0.3918	U	1 1/2	6	0.1067	1.3933	1 11/32
1/2	12	0.0534	0.4466	27/64	1 3/4	5	0.1281	1.6219	1 9/16
9/16	12	0.0534	0.5091	31/64	2	4 1/2	0.1423	1.8577	1 25/32
5/8	11	0.0582	0.5668	35/64	2 1/4	4	0.1601	2.0899	2
3/4	10	0.0640	0.6860	21/32	2 1/2	4	0.1601	2.3399	2 1/4
7/8	9	0.0711	0.8039	49/64					

BRITISH STANDARD FINE THREADS



P (pitch) $= \dfrac{1}{\text{no. of threads per inch}}$

D (depth) $= 0.6403P = \dfrac{.6403}{n}$

$H = 0.9605P$

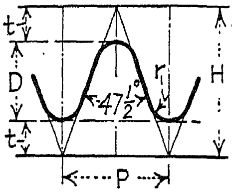
$\dfrac{H}{6} = 0.1600P$

$r = 0.1373P$

n = no. of threads per inch

Nominal diameter of screw	No. of threads per inch	Std. single depth of thread	Effective diameter (pitch diameter)	Commercial tap drill to produce 75% full thread	Nominal diameter of screw	No. of threads per inch	Std. single depth of thread	Effective diameter (pitch diameter)	Commercial tap drill to produce 75% full thread
1/4	26	0.0246	0.2254	No. 3	13/16	12	0.0534	0.7591	47/64
9/32	26	0.0246	0.2566	D	7/8	11	0.0582	0.8168	25/32
5/16	22	0.0291	0.2834	M	1	10	0.0640	0.9360	29/32
3/8	20	0.0320	0.3430	21/64	1 1/8	9	0.0711	1.0539	1 1/64
7/16	18	0.0356	0.4019	W	1 1/4	9	0.0711	1.1789	1 9/64
1/2	16	0.0400	0.4600	7/16	1 3/8	8	0.0800	1.2950	1 17/64
9/16	16	0.0400	0.5225	1/2	1 1/2	8	0.0800	1.4200	1 25/64
5/8	14	0.0457	0.5793	9/16	1 3/4	7	0.0915	1.6585	1 39/64
11/16	14	0.0457	0.6418	5/8	2	7	0.0915	1.9085	1 55/64
3/4	12	0.0534	0.6966	43/64					

TABLE 15.—BRITISH ASSOCIATION SCREW THREADS



Angle of thread $47\frac{1}{2}$ deg.

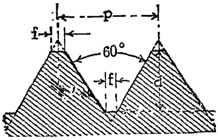
$$P \text{ (pitch)} = \frac{1}{\text{no. of threads per inch}}$$

D (depth) $= 0.6P$
$H = 1.136P$
$t = 0.268P$
$r = 0.182P$
n = no. of threads per inch

Seventy-five per cent of the full depth of thread is amply strong for all ordinary work.

Number	Diameter, mm. (major diameter)	Approximate diameter, inches	Pitch, mm.	Depth of thread, mm.	Effective diameter (pitch diameter), mm.	Core diameter (minor diameter), mm.	Drill sizes nearest commercial drill to produce 75 % depth of thread
0	6.0	0.236	1.00	0.600	5.400	4.80	No. 7
1	5.3	0.209	0.90	0.540	4.760	4.22	16
2	4.7	0.185	0.81	0.485	4.215	3.73	22
3	4.1	0.161	0.73	0.440	3.660	3.22	29
4	3.6	0.142	0.66	0.395	3.205	2.81	31
5	3.2	0.126	0.59	0.355	2.845	2.49	37
6	2.8	0.110	0.53	0.320	2.480	2.16	43
7	2.5	0.098	0.48	0.290	2.210	1.92	46
8	2.2	0.087	0.43	0.260	1.940	1.68	48
9	1.9	0.075	0.39	0.235	1.665	1.43	$\frac{1}{16}$ in.
10	1.7	0.067	0.35	0.210	1.490	1.28	54
11	1.5	0.059	0.31	0.185	1.315	1.13	56
12	1.3	0.051	0.28	0.170	1.130	0.96	59

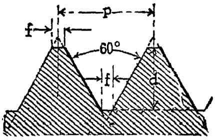
TABLE 16.—FRENCH (METRIC) STANDARD SCREW THREADS



Formula $\begin{cases} p = \text{Pitch} \\ d = \text{Depth} = p \times .64952 \\ f = \text{Flat} = \frac{p}{8} \end{cases}$

Diameter of Screw mm.	Pitch mm.	Diameter at Root of Thread mm.	Width of Flat mm.
3	0.5	2.35	.06
4	0.75	3.03	.09
5	0.75	4.03	.09
6	1.0	4.70	.13
7	1.0	5.70	.13
8	1.0	6.70	.13
8	1.25	6.38	.16
9	1.0	7.70	.13
9	1.25	7.38	.16
10	1.5	8.05	.19
11	1.5	9.05	.19
12	1.5	10.05	.19
12	1.75	9.73	.22
14	2.0	11.40	.25
16	2.0	13.40	.25
18	2.5	14.75	.31
20	2.5	16.75	.31
22	2.5	18.75	.31
22	3.0	18.10	.38
24	3.0	20.10	.38
26	3.0	22.10	.38
27	3.0	23.10	.38
28	3.0	24.10	.38
30	3.5	25.45	.44
32	3.5	27.45	.44
33	3.5	28.45	.44
34	3.5	29.45	.44
36	4.0	30.80	.5
38	4.0	32.80	.5
39	4.0	33.80	.5
40	4.0	34.80	.5
42	4.5	36.15	.56
44	4.5	38.15	.56
45	4.5	39.15	.56
46	4.5	40.15	.56
48	5.0	41.51	.63
50	5.0	43.51	.63
52	5.0	45.51	.63
56	5.5	48.86	.69
60	5.5	52.86	.69
64	6.0	56.21	.75
68	6.0	60.21	.75
72	6.5	63.56	.81
76	6.5	67.56	.81
80	7.0	70.91	.88

TABLE 17.—INTERNATIONAL STANDARD SCREW THREADS
Dimensions in millimeters



Formula $\begin{cases} p = \text{Pitch} \\ d = \text{Depth} = p \times .6495 \\ f = \text{Flat} = \frac{p}{\text{[illegible]}} \end{cases}$

Diam. of Screw	Pitch	Diam. of Screw	Pitch	Diam. of Screw	Pitch	Diam. of Screw	Pitch
6	1.00	18	2.50	39	4.00	68	6.00
7	1.00	20	2.50	42	4.50	72	6.50
8	1.25	22	2.50	45	4.50	76	6.50
9	1.25	24	3.00	48	5.00	80	7.00
10	1.50	27	3.00	52	5.00	88	7.50
11	1.50	30	3.50	56	5.50	96	8.00
12	1.75	33	3.50	60	5.50	116	9.00
14	2.00	36	4.00	64	6.00	136	10.00
16	2.00						

The "International Standard" is the same, with modifications noted, as that now in general use in France.

INTERNATIONAL STANDARD THREADS

At the "Congrès International pour l'Unification des Filetages," held in Zurich, October 24, 1898, the following resolutions were adopted:

"The Congress has undertaken the task of unifying the threads of machine screws. It recommends to all those who wish to adopt the metric system of threads to make use of the proposed system. This system is the one which has been established by the 'Society for the Encouragement of National Industries,' with the following modification adopted by this Congress.

"1. The clearance at the bottom of thread shall not exceed $\frac{1}{16}$ part of the height of the original triangle. The shape of the bottom of the thread resulting from said clearance is left to the judgment of the manufacturers. However, the Congress recommends rounded profile for said bottom.

"3. The table for Standard Diameters accepted is the one which has been proposed by the Swiss Committee of Action. [This table is given above.] It is to be noticed especially that 1.25 mm. pitch is adopted for 8 mm. diameter, and 1.75 mm. pitch for 12 mm. diameter. The pitches of sizes between standard diameters indicated in the table are to be the same as for the next smaller standard diameter."

TABLE 18.—THE METRIC SYSTEM OF MEASUREMENT

Measures of Length

1 millimeter (mm.) = 0.03937079 in., or about 1/25 in.
10 millimeters = 1 centimeter (cm.) = 0.3937079 in.
10 centimeters = 1 decimeter (dm.) = 3.937079 in.
10 decimeters = 1 meter (m.) = 39.37079 in., 3.2808992 ft., or 1.09361 yd.
10 meters = 1 decameter (Dm.) = 32.808992 ft.
10 decameters = 1 hectometer (Hm.) = 19.927817 rods
10 hectometers = 1 kilometer (Km.) = 1093.61 yd., or 0.621377 mi.
10 kilometers = 1 myriameter (Mm.) = 6.21377 mi.
1 inch = 2.54 cm., 1 foot = 0.3048 m., 1 yard = 0.9144 m., 1 rod = 0.5029 Dm., 1 mile = 1.6093 Km.

Measures of Weight

1 gramme (g.) = 15.4324874 gr. Troy, or 0.03215 oz. Troy, or 0.03527398 oz. avoir.
10 grammes = 1 decagramme (Dg.) = 0.3527398 oz. avoir.
10 decagrammes = 1 hectogramme (Hg.) = 3.527398 oz. avoir.
10 hectogrammes = 1 kilogramme (Kg.) = 2.20462125 lb.
1000 kilogrammes = 1 tonne (T.) = 2204.62125 lb. or 1.1023 tons of 2000 lb. or 0.9842 ton of 2240 lb. or 19.68 cwt.
1 grain = 0.0648 g., 1 ounce avoirdupois = 28.35 g., 1 pound = 0.4536 Kg., 1 ton 2000 lb. = 0.9072 T., 1 ton 2240 lb. = 1.016 T., or 1016 Kg.

Measures of Capacity

1 liter (l.) = 1 cubic decimeter = 61.0270515 cu. in., or 0.03531 cu. ft. or 1.0567 liquid qt. or 0.908 dry qt. or 0.26417 Amer. gal.
10 liters = 1 decaliter (Dl.) = 2.6417 gal., or 1.135 pk.
10 decaliters = 1 hectoliter (Hl.) = 2.8375 bu.
10 hectoliters = 1 kiloliter (Kl.) = 61027.0515 cu. in., or 28.375 bu.
1 cubic foot = 28.317 l., 1 gallon, Amer. = 3.785 l., 1 gallon, Brit. = 4.543 l.

TABLE 19.—METRIC CONVERSION TABLE

Millimeters	×	.03937	= Inches
Millimeters	=	25.400	× Inches
Meters	×	3.2809	= Feet
Meters	=	.3048	× Feet
Kilometers	×	.621377	= Miles
Kilometers	=	1.6093	× Miles
Square centimeters	×	.15500	= Square inches
Square centimeters	=	6.4515	× Square inches
Square meters	×	10.76410	= Square feet
Square meters	=	.09290	× Square feet
Square kilometers	×	247.1098	= Acres
Square kilometers	=	.00405	× Acres
Hectares	×	2.471	= Acres
Hectares	=	.4047	× Acres
Cubic centimeters	×	.061025	= Cubic inches
Cubic centimeters	=	16.3866	× Cubic inches
Cubic meters	×	35.3156	= Cubic feet
Cubic meters	=	.02832	× Cubic feet
Cubic meters	×	1.308	= Cubic yards
Cubic meters	=	.765	× Cubic yards
Liters	×	61.023	= Cubic inches
Liters	=	.01639	× Cubic inches
Liters	×	.26418	= U. S. gallons
Liters	=	3.7854	× U. S. gallons
Grams	×	15.4324	= Grains
Grams	=	.0648	× Grains
Grams	×	.03527	= Ounces, avoirdupois
Grams	=	28.3495	× Ounces, avoirdupois
Kilograms	×	2.2046	= Pounds
Kilograms	=	.4536	× Pounds
Kilograms per sq. cm	×	14.2231	= Lb. per sq. in.
Kilograms per sq. cm	=	.0703	× Lb. per sq. in.
Kilogram per cubic meter	×	.06243	= Lb. per cu. ft.
Kilogram per cubic meter	=	16.01890	× Lb. per cu. ft.
Metric tons (1000 kilograms)	×	1.1023	= Tons (2000 lb.)
Metric tons (1000 kilograms)	=	.9072	× Tons (2000 lb.)
Kilowatts	×	1.3405	= Horsepowers
Kilowatts	=	.746	× Horsepowers
Calories	×	3.9683	= B. T. units
Calories	=	.2520	× B. T. units
Francs	×	.193	= Dollars
Francs	=	5.18	× Dollars

By courtesy of *The American Machinist*, New York.

TABLE 20.—DECIMAL INCH EQUIVALENTS OF MILLIMETERS AND FRACTIONS OF MILLIMETERS[1]

Millimeter	Inches	Millimeter	Inches	Millimeter	Inches	Millimeter	Inches
1/100	.00039	33/100	.01299	64/100	.02520	95/100	.03740
2/100	.00079	34/100	.01339	65/100	.02559	96/100	.03780
3/100	.00118	35/100	.01378	66/100	.02598	97/100	.03819
4/100	.00157	36/100	.01417	67/100	.02638	98/100	.03858
5/100	.00197	37/100	.01457	68/100	.02677	99/100	.03898
6/100	.00236	38/100	.01496	69/100	.02717	1	.03937
7/100	.00276	39/100	.01535	70/100	.02756	2	.07874
8/100	.00315	40/100	.01575	71/100	.02795	3	.11811
9/100	.00354	41/100	.01614	72/100	.02835	4	.15748
10/100	.00394	42/100	.01654	73/100	.02874	5	.19685
11/100	.00433	43/100	.01693	74/100	.02913	6	.23622
12/100	.00472	44/100	.01732	75/100	.02953	7	.27559
13/100	.00512	45/100	.01772	76/100	.02992	8	.31496
14/100	.00551	46/100	.01811	77/100	.03032	9	.35433
15/100	.00591	47/100	.01850	78/100	.03071	10	.39370
16/100	.00630	48/100	.01890	79/100	.03100	11	.43307
17/100	.00669	49/100	.01929	80/100	.03150	12	.47244
18/100	.00709	50/100	.01969	81/100	.03189	13	.51181
19/100	.00748	51/100	.02008	82/100	.03228	14	.55118
20/100	.00787	52/100	.02047	83/100	.03268	15	.59055
21/100	.00827	53/100	.02087	84/100	.03307	16	.62992
22/100	.00866	54/100	.02126	85/100	.03346	17	.66929
23/100	.00906	55/100	.02165	86/100	.03386	18	.70866
24/100	.00945	56/100	.02205	87/100	.03425	19	.74803
25/100	.00984	57/100	.02244	88/100	.03465	20	.78740
26/100	.01024	58/100	.02283	89/100	.03504	21	.82677
27/100	.01063	59/100	.02323	90/100	.03543	22	.86614
28/100	.01102	60/100	.02362	91/100	.03583	23	.90551
29/100	.01142	61/100	.02402	92/100	.03622	24	.94488
30/100	.01181	62/100	.02441	93/100	.03661	25	.98425
31/100	.01220	63/100	.02480	94/100	.03701	26	1.02362
32/100	.01260						

[1] *Courtesy* of Brown & Sharpe Mfg. Company.

TABLE 21.—DECIMAL EQUIVALENTS OF THE NUMBER AND LETTER SIZES OF TWIST DRILLS

No.	Size in decimals	No.	Size in decimals	No.	Size in decimals	No.	Size in decimals
1	.2280	21	.1590	41	.0960	61	.0390
2	.2210	22	.1570	42	.0935	62	.0380
3	.2130	23	.1540	43	.0890	63	.0370
4	.2090	24	.1520	44	.0860	64	.0360
5	.2055	25	.1495	45	.0820	65	.0350
6	.2040	26	.1470	46	.0810	66	.0330
7	.2010	27	.1440	47	.0785	67	.0320
8	.1990	28	.1405	48	.0760	68	.0310
9	.1960	29	.1360	49	.0730	69	.02925
10	.1935	30	.1285	50	.0700	70	.0280
11	.1910	31	.1200	51	.0670	71	.0260
12	.1890	32	.1160	52	.0635	72	.0250
13	.1850	33	.1130	53	.0595	73	.0240
14	.1820	34	.1110	54	.0550	74	.0225
15	.1800	35	.1100	55	.0520	75	.0210
16	.1770	36	.1065	56	.0465	76	.0200
17	.1730	37	.1040	57	.0430	77	.0180
18	.1695	38	.1015	58	.0420	78	.0160
19	.1660	39	.0995	59	.0410	79	.0145
20	.1610	40	.0980	60	.0400	80	.0135

LETTER SIZES OF DRILLS

Letter	Size in decimals	Letter	Size in decimals
A 15/64	.234	N	.302
B	.238	O 5/16	.316
C	.242	P 21/64	.323
D	.246	Q	.332
E 1/4	.250	R 11/32	.339
F	.257	S	.348
G	.261	T 23/64	.358
H 17/64	.266	U	.368
I	.272	V 3/8	.377
J	.277	W 25/64	.386
K 9/32	.281	X	.397
L	.290	Y 13/32	.404
M 19/64	.295	Z	.413

B